SVSLOPE™

2D Slope Stability Modeling Software

Tutorial Manual

Written by:
Murray Fredlund, Ph.D.
Tiequn Feng, Ph.D.
Robert Thode, B.Sc.G.E.

Edited by:
Murray Fredlund, Ph.D.

SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada

1   Introduction

The Tutorial Manual serves a special role in guiding the first time users of the SVSLOPE software through a typical example problem. The example is "typical" in the sense that it is not too rigorous on one hand and not too simple on the other hand.

In particular this tutorial manual is designed to guide users through the range of reasonable models which may be encountered in typical slope stability modeling. The following examples represent the most typical models encountered in the traditional slope stability modeling practice and therefore include:

1. Basic Slope,

2. Weak Layer Example,

3. Geomembrane Example,

4. Dynamic Programming Example,

5. Kulhawy Method,

6. 3D Multi Planar Example,

7. 3D Submergence Example,

8. 3D General Sliding Surface,

9. 2D Rapid Drawdown Example,

10. 3D Rapid Drawdown Example,

11. Hong Kong Example, and

12. 3D Arbitrary Sliding Direction

2   Authorization

Certain features in SVOFFICE are not available in the STUDENT, version of the software. Perform the following steps to check if STANDARD, PROFESSIONAL, or ELITE authorization is activated:

1. Plug in the USB security key,

2. Select *File > Authorization...* from the menu on the SVOFFICE Project Manager,

3. The software will display the authorization under the *Level Authorized* heading. If not, the security codes provided by SoilVision Systems at the time of purchase have not yet been entered. Please see the Authorization section of the SVOFFICE User's Manual for instructions on entering these codes.

3   Basic Slope

The following example will introduce some of the features included in SVSLOPE and will set up a model using limited equilibrium method of slices and the Grid and Radius search method for circular slip surfaces. The purpose of this model is to determine the factor of safety of a simple model. The model dimensions and material properties are in the next section.

This example consists of a simple slope with two layers and a water table. The problem is analyzed using the Bishop Simplified method as well as the Morgenstern-Price method. The purpose of this example is to illustrate the calculation of the factor of safety for a simple slope example.

This original model can be found under:

Project: Slopes_Group_2
Model: VW_9

Minimum authorization required to complete this tutorial: STUDENT

Model Description and Geometry

3.1   Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

   a.   Create model
   b.   Specify analysis settings
   c.   Enter geometry
   d.   Specify search method geometry
   e.   Specify pore water pressure

    f. Apply material properties

    g. Run model

    h. Visualize results

The details of these outlined steps are given in the following sections.

> **N O T E :**
> Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,

2. Press the Clear Search button (if enabled),

3. Create a new project called "UserTutorial" by pressing the *New* button next to the list of projects,

4. Create a new model called "Basic Slope" by pressing the *New* button next to the list of models. Use the settings below when creating this new model:

    Application: SVSLOPE

    Model Name: Basic Slope

    Units: Metric

    Slip Direction: Left to Right

5. Click on the *World Coordinate System* tab,

6. Enter the World Coordinates System coordinates shown below into the dialog,

    x min = -2   x max = 40

    y min = -2   y max = 25

7. Click on *OK*.

The new model will be automatically added under the recently created UserTutorial project.

SVSLOPE now opens to show a grid and the Options dialog (*View > Options*) pops up. Click *OK* to accept the default horizontal and vertical grid spacing.

b. Specify Analysis Settings (Model > Settings)

In SVSlope the Analysis Settings provide the information for what model output will be available in ACUMESH. These settings will be specified as follows:

1. Select *Model > Settings...* from the menu,

2. Move to the *Slip Surface* tab and notice that the following items are selected:

    Slip Direction: *Left to Right*
    Slip Shape: *Circular*
    Search Method: *Grid and Tangent*

3. Select the *Calculation Methods* tab from the dialog and select the method types as shown below:

> Bishop Simplified
> Spencer
> M-P
> GLE

4. Press *OK* to close the dialog.

c. **Enter Geometry** (Model > Geometry)

Model geometry is defined as a set of regions and a series of layers. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from either .DXF files or from existing models.

This model will be divided into two regions, which are named R1 and R2. Each region will have one of the materials specified as its material properties. The shapes that define each material region will now be created. Note that when drawing a geometric shape, information will be added to the region that is current in the Region Selector. The Region Selector is at the top of the workspace.

- **Define R1 Region**

    1. Select *Draw > Geometry > Region Polygon* from the menu,

    2. The cursor will now be changed to a cross hair,

    3. Move the cursor near (0,9) in the drawing space. You can view the coordinates of the current position of the mouse in the status bar,

    4. To select the point as part of the shape left click on the point,

    5. Now move the cursor near (0,14) and left-click the mouse. A line is now drawn from (0,9) to (0,14),

    6. Now move the cursor near (10,14) and left-click the mouse.

    7. Move the cursor near the point (20,9). Double click on the point to finish the shape. A line is now drawn from (0,9) to (20,9) and the shape is automatically finished by SVSLOPE by drawing a line from (20,9) back to the start point, (0,9).

Repeat this process to define the R2 region according to the data provided in the table below.

Region: R1

X (m)	Y (m)
0	9
0	14
10	14
20	9

Region: R2

X (m)	Y (m)
0	0
0	9
20	9
30	4
40	4
40	0

N O T E :

If an error is made when entering the region geometry the user may recover from the error and start again by one of the following methods:
   a. Press the escape (esc) key.
   b. Select a region shape and press the delete key.
   c. Use the Undo function on the Edit menu.

If all model geometry has been entered correctly the shape should look like the diagram at the start of this tutorial.

d. Specify Search Method Geometry

The Grid and Tangent method of searching for the critical slip surface has already been selected in the previous step. Now the user must draw the graphical representation of the grid and tangent objects on the screen. This is accomplished through the following steps:

GRID

1.   Select *Model > Slip Surface > Grid and Tangent*,
2.   Select the *Grid* tab,
3.   Enter the values for the grid as specified in the table below (the grid values may also be drawn on the CAD window),
4.   Move to entering the tangent values.

X (m)	Y (m)
23	25
22	19
26	19

X increments: 2
Y increments: 3

TANGENT

1.   Select the *Tangent* tab,
2.   Enter the values for the tangent as specified in the table below (the grid values may also be drawn on the CAD window),

3.   Press *OK* to close the dialog.

X (m)	Y (m)
15	4
15	2
29	2
29	4

Radius Increments: 2

The grid and tangent graphics should now be displayed on the CAD window.

e. Specify Pore Water Pressure (Model > Pore Water Pressure)

A water table or a piezometric line must be specified as an initial condition for this model. In this model a piezometric line will be used. In order to specify that a piezometric line will be entered the user needs to following these steps:

1.   Select *Model > Pore Water Pressure > Settings...*,

2.   Select "Water Surfaces" as the Pore-Water Pressure Method,

3.   Press *OK* to close the dialog.

The user must then proceed to graphically enter the piezometric line:

1.   Select *Model >Pore Water Pressure > Piezometric Line*,

2.   Under the *Points* tab, click on the *New Line* button,

3.   Enter in the *X* and *Y* coordinates as provided in the table below,

4.   Press *OK* to close the dialog.

X (m)	Y (m)
0	11
15	8
30	3
40	3

f. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the two materials that will be used in the model. R1 region will have the Upper Soil applied to it and R2 will have the Lower Soil applied. This section will provide instructions on creating the Upper Soil. Repeat the process to add the second material.

1.   Open the *Materials* dialog by selecting *Model > Materials > Manager...* from the menu,

2.   Click the *New...* button to create a material,

3.   Enter "Upper Soil" for the material name in the dialog that appears and choose

Mohr Coulomb for the Shear Strength type of this material,

4. Press *OK* to close the dialog. The *Material Properties* dialog will open automatically,

> **N O T E :**
> When a new material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned to it will display that material's fill color.

5. Move to the *Shear Strength* tab and enter the parameter values given in the table below,
6. Click the *OK* button to close the *Material Properties* dialog,
7. Repeat these steps to create the Lower Soil material,
8. Press OK to close the Materials Manager.

Material	Shear Strength Type	Unit Weight (kN/m³)	Cohesion (kPa)	Friction Angle phi (deg)
Upper Soil	Mohr Coulomb	15	5	20
Lower Soil	Mohr Coulomb	18	10	25

Once all material properties have been entered, we must apply the materials to the corresponding regions.

1. Open the *Region Properties* dialog by selecting *Model > Geometry > Region Properties* from the menu,
2. Select the *R1* region and assign the *Upper Soil* material to this region,
3. Select the R2 region and assign the *Lower Soil* material to this region,
4. Press the *OK* button to accept the changes and close the dialog.

g. Run Model   (Solve > Analyze)

The next step is to analyze the model.

1. Select *Solve > Analyze* from the menu. A pop-up dialog will appear and the solver will start,
2. Press the *OK* button to close the dialog.

h. Visualize Results   (Window > AcuMesh)

After the model has been run, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on *Yes*. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH as appears in the diagrams following. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen. To switch

between the results of the different methods selected, click on the drop down menu (as shown below) at the top of the screen and select the method you would like to view.

Spencer

3.2   Results and Discussions

If the model has been appropriately entered into the software the approximate following results should be shown for the Bishop method. The user may display results from different methods by clicking the combo box on the display which lists the different analysis methods (Bishop, Spencer, etc.). It should be noted that it is typically recommended that the search grid of centers be somewhat larger in order to ensure that a critical center is not missed.
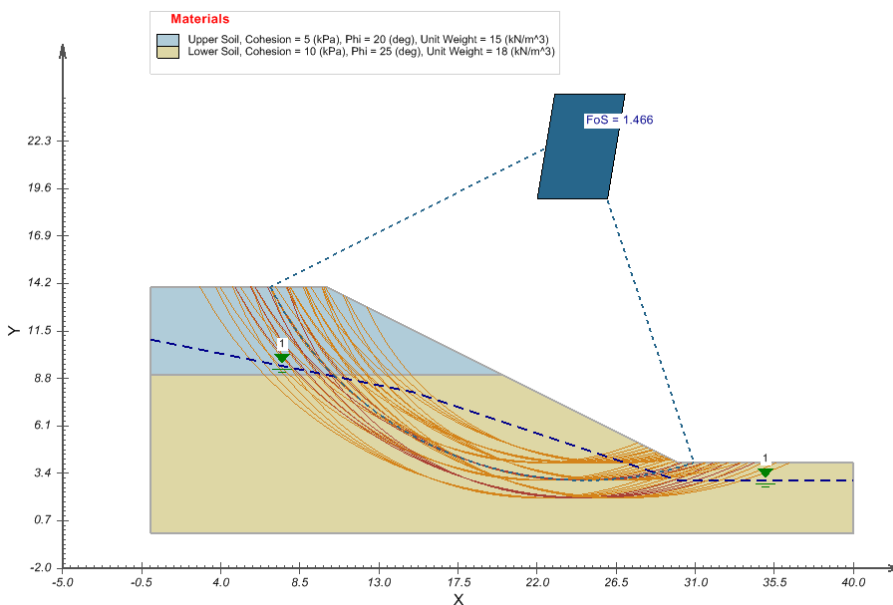
The correct results for this example are:

Method	SVSLOPE	
	Moment	Force
Bishop Simplified	1.466	
Spencer	1.469	1.469
M-P	1.468	1.467
GLE	1.467	1.467

4   Weak Layer Example

This is a more complex example involving a weak layer, pore-water pressures and surcharges. The ACADS verification program received a wide range of answers for this model and fully expected this during the program. The soil parameters, external loadings and piezometric surface are shown in the following diagram. Tension cracks are ignored in this example. The model requirement is that the noncircular slip surface and the corresponding factor of safety are required.

This original model can be found under:

Project: Slopes_Group_1
Model: VS_9

Minimum authorization required to complete this tutorial: STANDARD

Model Description and Geometry

A block search for the critical noncircular failure surface is carried out by defining two line searches to block search squares within the weak layer. A number of different random surfaces were generated by the search and the results compared well with the actual results.
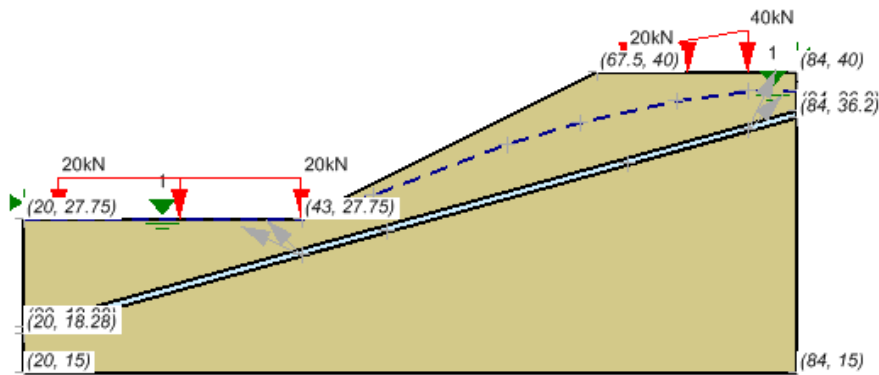
Region: R1

x (m)	y (m)
20	27.75
20	18.88
84	36.8
84	40
67.5	40
43	27.75

Region: R2

x (m)	y (m)
20	18.88
20	18.28
84	36.2

84	36.8

Region: R3

x (m)	y (m)
20	18.28
20	15
84	15
84	36.2

Material Properties

Material	c (kPa)	φ (degrees)	γ (kN/m³)
Soil #1	28.5	20.0	18.84
Soil #2	0.0	10.0	18.84

Piezometric Line

Pt. #	Xc (m)	Yc (m)
1	20.0	27.75
2	43.0	27.75
3	49.0	29.8
4	60.0	34.0
5	66.0	35.8
6	74.0	37.6
7	80.0	38.4
8	84.0	38.4

Loading   Type: Trapezoid

X (m)	Y (m)	Normal Stress (kN/m²)
23.00	27.75	20.00
43.00	27.75	20.00
70.00	40.00	20.00
80.00	40.00	40.00

Block Search Parameters

Left Block

43	24.807
43	24.807
50	26.769

X increments: 10
Y increments: 1
Start Angle: 135 degrees
End Angle: 155 degrees
Left Increments: 2

Right Block

70	32.376
70	32.376
80	35.179

X increments: 10
Y increments: 1
Start Angle: 45 degrees
End Angle: 65 degrees
Right Increments: 2

4.1 Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

     a. Create model

     b. Specify analysis settings

     c. Enter geometry

     d. Specify search method geometry

     e. Specify pore water pressure

     f. Specify loading conditions

     g. Apply material properties

     h. Run model

     i. Visualize results

The details of these outlined steps are given in the following sections.

> **N O T E :**
> Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,

2. Press the Clear Search button (if enabled),

3. Create a new project called UserTutorial by pressing the *New* button next to the list of projects,

4. Create a new model called "Weak Layer Example" by pressing the *New* button next to the list of models. Use the settings below when creating this new model:

       Application: SVSLOPE

       Model Name: Weak Layer Example

       Units: Metric

Slip Direction: Right to Left

5. Click on the *World Coordinate System* tab,

6. Enter the World Coordinates System coordinates shown below into the dialog (leave Global Offsets as zero),

        x min = 15   x max = 90

        y min = 10   y max = 60

7. Click on *OK*.

The new model will be automatically added under the UserTutorial project. SVSLOPE now opens to show a grid and the Options dialog (*View > Options)* pops up. Click *OK* to accept the default horizontal and vertical grid spacing of 0.5.

b. Specify Analysis Settings (Model > Settings)

In SVSlope the Analysis Settings provide the information for what model output will be available in ACUMESH. These settings will be specified as follows:

1. Select *Model > Settings* from the menu,

2. Move to the *Slip Surface* tab and ensure that the following items are selected:

      Slip Direction: *Right to Left*

      Slip Shape: *Non-Circular*

      Search Method: *Block*

3. Select the *Calculation Methods* tab from the dialog and select the method types as shown below:

      Spencer

      GLE

4. For GLE method, press the *Lambda...* button,

5. Enter a Start Value of -1.25, an Interval of 0.25, and a Number of 11,

6. Press the *Generate* button,

7. Press OK to close the dialogs.

c. Enter Geometry (Model > Geometry)

Model geometry is defined as a set of regions. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from either .DXF files or from existing models.

The shapes that define each material region will now be created.

- **Define R1 Region**

1. Ensure the R1 region is current in the region selector. The region selector appears underneath the menus at the top of the screen,

2. Select *Draw > Geometry > Polygon Region* from the menu,

3. The cursor will now be changed to cross hairs,

4. Move the cursor near (20,27.75) in the drawing space. You can view the coordinates of the current position the mouse is at in the status bar just above the command line,

5. When the cursor is near the point left-click. This will cause the cursor to snap to the point (The SNAP and GRID options in the status bar must both be on),

6. Now move the cursor near (20,18.88) and left-click. A line is now drawn from (20,27.75) to (20,18.88),

7. In the same manner then enter the following points:

   (84,36.8)

   (84,40)

   (67.5,40)

   (43,27.75)

8. Move the cursor near the point (43,27.75) and double-click on the point to finish the shape.

Repeat this process to define the R2 and R3 regions according to the information provided at the start of this tutorial.

> **N O T E :**
> If an error is made when entering the region geometry the user may recover from the error and start again by one of the following methods:
>   a. Press the escape (esc) key
>   b. Select a "Region Shape" and press the *delete* key
>   c. Use the Undo function on the Edit menu

If all model geometry has been entered correctly the shape should look like the diagram at the beginning of this tutorial.

d. Specify Search Method Geometry

This particular model makes use of a block search methodology. The block search parameters may be entered through the following steps:

1. Open the *block search* dialog through the *Model > Slip Surface > Block Search*... menu option,

2. Enter the right block search data and then left block search data as specified in the start of this tutorial,

3. Click *OK* to close the dialog.

e. Specify Pore Water Pressure (Model > Pore Water Pressure)

Generally information will be entered either for a water table or a piezometric line. In this model a piezometric line will be used. In order to specify that a piezometric line will be entered the user needs to following these steps:

1. Select *Model > Pore Water Pressure > Settings*...,

2. Select "Water Surfaces" as the Pore-Water Pressure Method,

3.   Press *OK* to close the dialog.

The user must then proceed to graphically enter the piezometric line:

1.   Select *Model > Pore Water Pressure > Piezometric Line*,

2.   Under the *Points* tab, click on the *New Line* button,

3.   Enter in the *X*(m) and *Y*(m) co-ordinates as provided at the start of this tutorial,

4.   Press *OK* to close the dialog.

f.   Specify Loading Conditions

Two distributed loads are applied in this numerical model. The instructions for applying these distributed loads are as follows:

1.   Select *Draw > Loading > Distributed Load*, then

2.   Enter the data as provided in the start of this tutorial,

3.   Click *OK* to close the dialog. You will need to do this for each load separately.

g.   Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the two materials that will be used in the model. R1 region will have the Upper Soil applied to it and R2 will have the Lower Soil applied to it. This section will provide instructions on creating the Upper Soil. Repeat the process to add the second material.

1.   Open the *Materials* dialog by selecting *Model > Materials > Manager* from the menu,

2.   Click the *New* button to create a material,

3.   Enter "Upper Soil" for the material name in the dialog that appears and choose Mohr Coulomb for the Shear Strength type of this material,

4.   Press *OK* to close the dialog. The *Material Properties* dialog will open automatically,

> **N O T E :**
> When a new material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned to it will display that material's fill color.

5.   Move to the *Shear Strength* tab,

6.   Enter the Unit Weight value of 18.84 kN/m^3,

7.   Enter the Cohesion, c: value of 28.5 kPa,

8.   Enter the Friction Angle, phi value of 20.0 degrees,

9.   Click the *OK* button to close the *Shear Strength* dialog,

10. Repeat these steps to create the Lower Soil material using the information provided at the beginning of the tutorial.

Once all material properties have been entered, we must apply the materials to the corresponding regions.

1. Open the *Region Properties* dialog by selecting *Model > Geometry > Region Properties* from the menu,

2. Select the R1 region and assign the Upper Soil material to this region,

3. Select the R2 region and assign the Lower Soil material to this region,

4. Select the R3 region and assign the Upper Soil material to this region,

5. Press the *OK* button to accept the changes and close the dialog.

h. Run Model (Solve > Analyze)

The next step is to analyze the model.

1. Select *Solve > Analyze* from the menu. A pop-up dialog will appear and the solver will start,

2. Press the OK button to close the dialog.

i. Visualize Results (Window > AcuMesh)

After the model has been run, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on *Yes*. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH as appears in the diagrams following. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen. To switch between the results of the different methods selected, click on the drop down menu (as shown below) at the top of the screen and select the method you would like to view.

4.2   Results and Discussions

After the model is completed the user may view the results in the ACUMESH software by pressing the open ACUMESH icon on the process toolbar. The results will contain all trial slip surfaces as well as the most critical slip surface results. In order to identify the most critical slip surface the user may perform the following steps:

1. Select "Slip Surfaces" from the menu item Slips, and

2. Click the *Show Trial Slip Surfaces* button, this will cause all the trial slip surfaces to not be displayed.

The user may also plot the slices used in the analysis of the critical slip surfaces through the slips show slices menu option. The information on any particular slice may be displayed through the slips slice information dialog. A slice information dialog will appear and the user may click on a new particular slice on the slope to display the details of that slice. The analysis results in a factor of safety of 0.741 for the Spencer method (shown below) and 0.708 for the GLE method.

Materials

	Soil 1	Cohesion = 28.5 (kPa)	Phi = 20 (deg)	Unit Weight = 18.84 (kN/m^3)
	Soil 2	Cohesion = 0 (kPa)	Phi = 10 (deg)	Unit Weight = 18.84 (kN/m^3)

Spencer's Method
Search Method: Block
FOS: 0.741
Total Mass: 4.082E+003 kPa
Total Volume: 2.167E+002 m^3
Total Resisting Moment: 3.754E+004 kNm
Total Activating Moment: 5.063E+004 kNm
Total Resisting Force: 7.551E+002 kN
Total Activating Force: 1.019E+003 kN
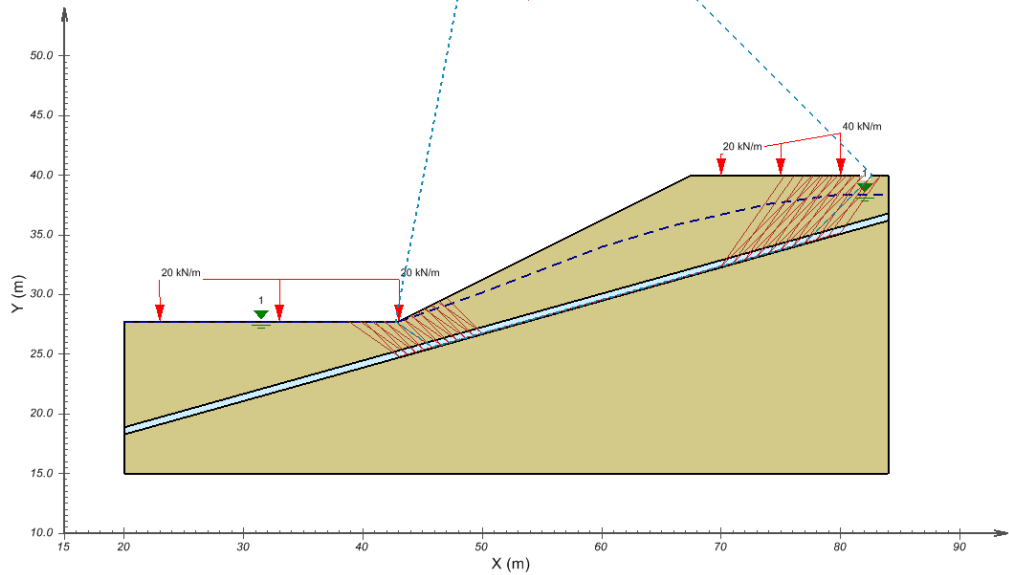
5   Geomembrane Example

The following example will introduce some of the features included in SVSLOPE and will set up a model using the limit equilibrium method of slices and the Grid and Radius search method for circular slip surfaces. The purpose of this model is to determine the effects of reinforcements. The model dimensions and material properties are in the next section.

This original model can be found under:
Project: Slopes_Group_2
Model: VW_6_Fabric

Minimum authorization required to complete this tutorial: STANDARD

Model Description and Geometry
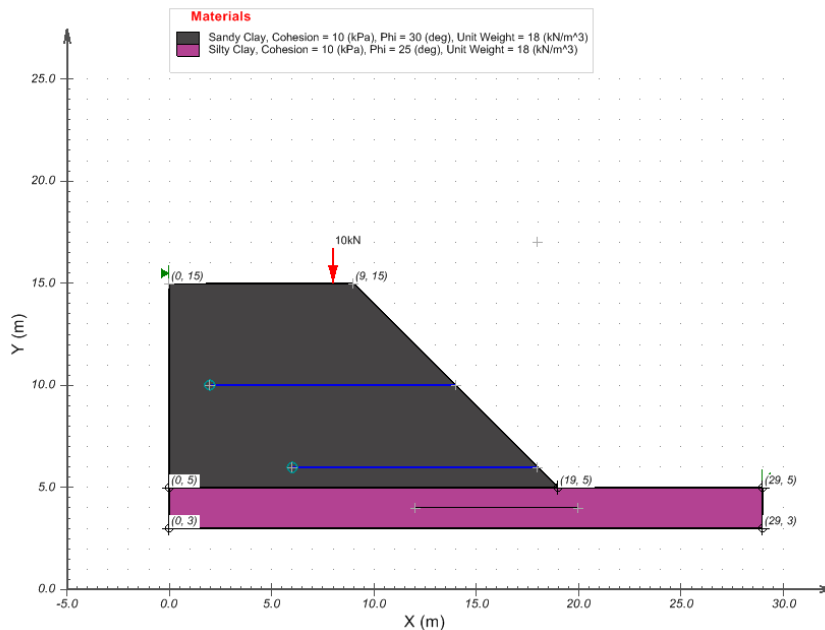
Region Geometries

Region: R1

x (m)	y (m)
0	5
0	15
9	15
19	5

Region: R2

x (m)	y (m)
0	3

0	5
19	5
29	5
29	3

Material Properties

Sandy Clay

Shear Strength Type	Unit Weight kN/m^3	Cohesion, c: kPa	Friction Angle, phi: deg
Mohr Coulomb	18	10	30

Silty Clay

Shear Strength Type	Unit Weight kN/m^3	Cohesion, c: kPa	Friction Angle, phi: deg
Mohr Coulomb	18	10	25

Grid and Tangent

Grid - Points

	X	Y
Upper Left	18	17
Lower Left	18	17
Lower Right	18	17

Tangent - Points

	X	Y
Upper Left	12	4
Lower Left	12	4
Lower Right	20	4
Upper Right	20	4

X increments   Increments

Y increments

Loading

Line Load #1

Orientation: Vertical

	Start Point		End Point	
Magnitude	10	kN		kN
X-coord:	8	m		m
Y-Coord:	15	m		m

Analysis Settings

Pore-Water Pressure

Pore Fluid Unit Weight   9.807   kN/m^3

LE Convergence Options

Number of Slices	Tolerance	Max. no. of iterations
30	0.0010	50

5.1   Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

 a. Create model

 b. Specify analysis settings

 c. Enter geometry

 d. Specify search method geometry

 e. Specify pore water pressure

 f. Specify loading conditions

 g. Apply material properties

 h. Add Supports

 i. Run model

 j. Visualize results

The details of these outlined steps are detailed in the following sections.

> **N O T E :**
> Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

The following steps are required to create the model:

 1. Open the *SVOFFICE Manager* dialog,

 2. Press the Clear Search button (if enabled),

 3. Create a new project called UserTutorial by pressing the *New* button next to the list of projects,

 4. Create a new model called VW_6_Tutorial by pressing the *New* button next to the list of models. The new model will be automatically added under the recently created UserTutorial project. Use the settings below when creating this new model,

 5. Select the following:

   Application: SVSLOPE

   System: 2D

   Units: Metric

   Slip Direction: Left to Right

6. Click on the *World Coordinate System* tab,

7. Enter the World Coordinates System coordinates shown below into the dialog,

>   x min = -2   x max = 30

>   y min = 2   y max = 18

6. Click *OK* to close the dialog,

7. Click *OK* to accept the default Options dialog settings.

b. Specify Analysis Settings (Model > Settings)

In SVSlope the Analysis Settings provide the information for what type of analysis will be performed. These settings will be specified as follows:

1. Select *Model > Settings* from the menu,

2. Select the Slip Surface tab,

>   Slip Direction: Left to Right

>   Slip Shape: Circular

>   Search Method: Grid and Tangent

3. Select the *Calculation Methods* tab from the dialog and select the method types as shown below:

>   Ordinary / Fellenius

>   Bishop Simplified

>   Janbu Simplified

>   Morgenstern-Price

>   GLE (Fredlund)

4. For GLE method, press the *Lambda...* button,

5. Enter a Start Value of -1.25, an Interval of 0.25, and a Number of 11,

6. Press the *Generate* button,

7. Press *OK* to close the dialogs.

c. Enter Geometry (Model > Geometry)

Model geometry is defined as a set of regions. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from either .DXF files or from existing models.

This model will be divided into two regions, which are named R1 and R2. The shapes that define each material region can be created by the following steps.

- **Define R1 Region**

1. Select *Draw > Geometry > Region Polygon* from the menu,

2. Move the cursor near (0,5) in the drawing space. You can view the coordinates of the current position the mouse is at in the status bar just above the command line,

3. Continue drawing the following points in order, and

x (m)	y (m)
0	5
0	15
9	15
19	5

4.　Double click on the final point to finish the shape.

Repeat this process to define the R2 region according to the information provided at the start of this tutorial.

> **N O T E :**
>
> If an error is made when entering the region geometry the user may recover from the error and start again by one of the following methods:
> > a. Press the escape (esc) key.
> > b. Select a region shape and press the *delete* key.
> > c. Use the Undo function on the Edit menu.

If all model geometry has been entered correctly the shape should look like the diagram at the beginning of this tutorial.

d. Specify Search Method Geometry

The Grid and Tangent method of searching for the critical slip surface has already been selected in the previous step. Now the user must draw the graphical representation of the grid and tangent objects on the screen. This is accomplished through the following steps:

GRID

1.　Select *Model > Slip Surface > Grid and Tangent,*
2.　Select the *Grid* tab,
3.　Enter the values for the grid as specified at the start of this tutorial (the grid values may also be drawn on the CAD window),
4.　Move to entering the tangent values.

TANGENT

1.　Select the *Tangent* tab,
2.　Enter the values for the tangent as specified at the start of this tutorial (the grid values may also be drawn on the CAD window),
3.　Close the dialog.

The grid and tangent graphics should now be displayed on the CAD window.

e. Specify Pore Water Pressure (Model > Pore Water Pressure)

There are no initial conditions associated with this tutorial.

f.　Specify Loading Conditions

There is a single line-load used for the current model. The following steps are required in order to apply this line load to the current model.

1. Open the *Line Load* dialog by selecting *Model > Loading > Line Load* from the menu,
2. Click the *New* button to create a new line load object,
3. Enter a value of 10 kN for the magnitude,
4. Enter a value of 8 m for the *X* coordinate,
5. Make sure that the load has a "Vertical" orientation,
6. Click OK to close the dialog.

g. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the two materials that will be used in the model. *R1* region will have the *Sandy Clay* applied to it and *R2* will have the *Silty Clay* applied to it. This section will provide instructions on creating the Sandy Clay. Repeat the process to add the second material.

1. Open the *Materials* dialog by selecting *Model > Materials > Manager* from the menu,
2. Click the *New* button to create a material,
3. Enter "Sandy Clay" for the material name in the dialog that appears and choose Mohr Coulomb for the Shear Strength type of this material,
4. Press *OK* to close the dialog. The *Material Properties* dialog will open automatically,

> **N O T E :**
> When a new material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned to it will display that material's fill color.

5. Move to the *Shear Strength* tab,
6. Enter the Unit Weight value of 18 kN/m^3,
7. Enter the Cohesion, c: value of 10 kPa,
8. Enter the Friction Angle, phi value of 30 degrees,
9. Click the OK button to close the Shear Strength dialog, and
10. Repeat these steps to create the Silty Clay material using the information provided at the beginning of the tutorial.

Once all material properties have been entered, we must apply the materials to the corresponding regions.

1. Open the Region Properties dialog by selecting *Model > Geometry > Regions* from the menu,
2. Select the *R1* region and assign the *Sandy Clay* material to this region,
3. Select the *R2* region and assign the *Silty Clay* material to this region,

4.   Press the OK button to accept the changes and close the dialog.

h. Add Supports (Model > Support)

The next step is to analyze the model.

1.   Open the *Support Type Manager* dialog by selecting *Model > Support > Type Manager* from the menu,

2.   Press the *New* button to open the *New Support Property* dialog,

3.   Select Geotextile as the *Support Type* and enter a name,

4.   Click *OK,*

5.   Select Passive as the Force Application,

6.   Set 0 kPa for Adhesion,

7.   Click *OK,* to close the Support Type Manager,

8.   Open the *Support Geometry* dialog by selecting *Model > Support > Geometry* from the menu,

9.   Click New to create a new support entry,

10. Leave the Orientation as None and enter the coordinates (18,6) and (6,6),

11. Select Geotextile as the *Property Name,*

12. Click New to create a second support entry,

13. Leave the Orientation as None and enter the coordinates (14,10) and (2,10),

14. Select Geotextile as the *Property Name,*

15. Click *OK,* to close the dialog.

i. Run Model (Solve > Analyze)

The next step is to analyze the model.

16. Select *Solve > Analyze* from the menu. A pop-up dialog will appear,

17. Click on the *green arrow* button on the bottom of the dialog to start the solver. This action will finish the calculations and save the results,

18. Click on the *Close* button to close the dialog.

j. Visualize Results (Window > AcuMesh)

After the model has been run, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on *Yes*. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH as appears in the diagrams following. To switch back and forth between your original geometry & ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen. To switch between the results of the different methods selected, click on the drop down menu (as shown below) at the top of the screen and select the method you would like to view.

Spencer

5.2   Results and Discussions

The results of the calculation of the factor of safety and the critical slip surface for the Ordinary Method are shown below. At the end of calculation the factor of safety is a result approximately 1.525. The support force distribution in shown along each support in the screenshot below from 0 to 50.
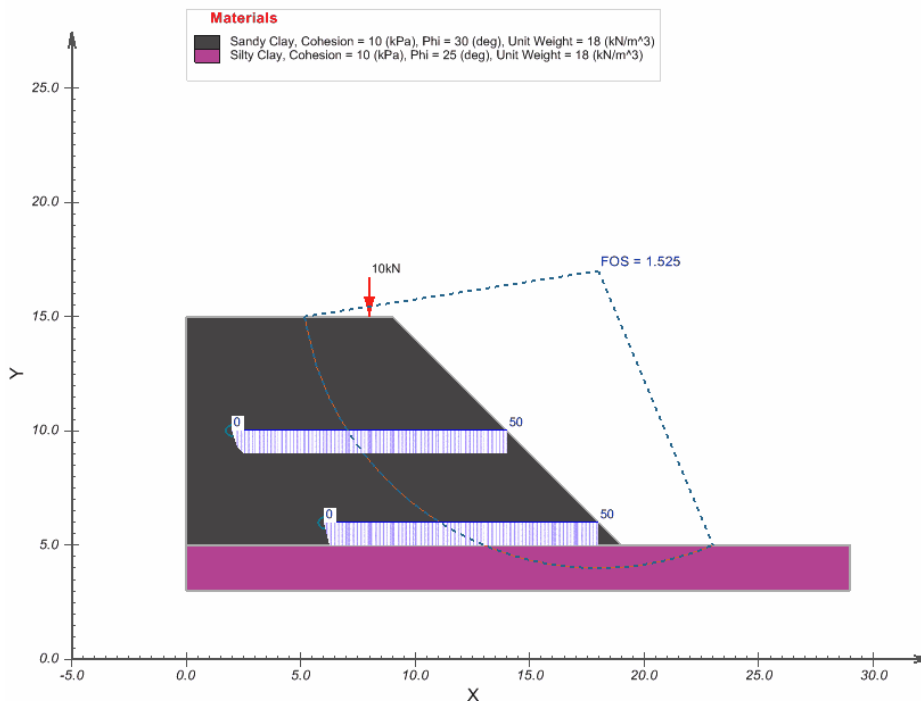
The correct results for this example are:

Method	SVSLOPE	
	Moment	Force
Ordinary	1.525	
Bishop Simplified	1.667	
Janbu Simplified		1.509
M-P	1.646	1.645
GLE	1.646	1.646

6   Dynamic Programming Example

This example will introduce the user to modeling in SVSLOPE using the SAFE-DP stress-based limit equilibrium calculation method and the Dynamic Programming search method for non-circular slip surfaces. The purpose of this model is to determine the factor of safety of a simple model. The model dimensions and material properties are in the next section.

This example consists of a three-layer slope with a thin weak layer. A combination of SVSolid and SVSlope are used to solve this model. The purpose of this example is to illustrate the calculation of the factor of safety for a the slope.

This original model can be found under:

Project: Slopes_STUDENT
Model: SAFE_52_EDU

Minimum authorization required to complete this tutorial: PROFESSIONAL

Model Description and Geometry
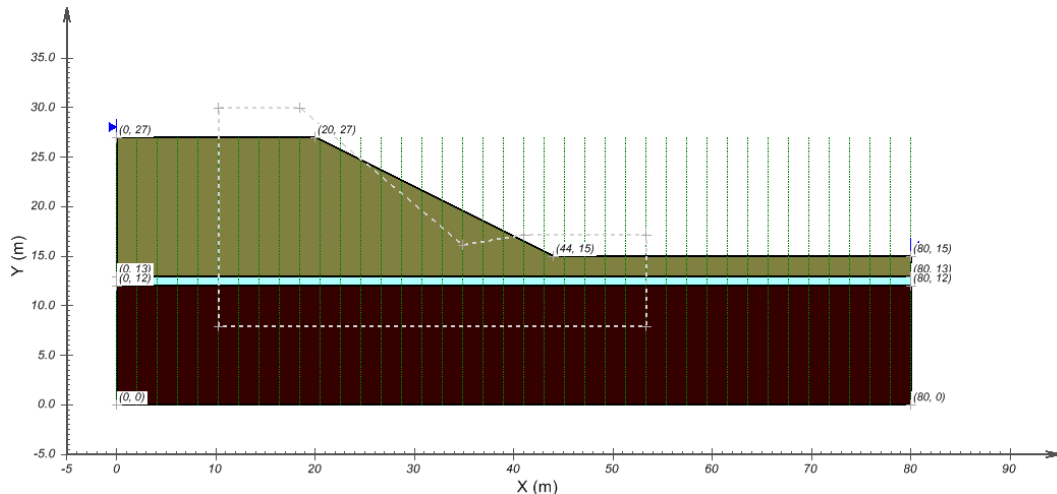
6.1   Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

SVSolid Steps:

       a.   Create model
       b.   Enter geometry
       c.   Specify boundary conditions
       d.   Apply SVSolid material properties
       e.   Combine SVSlope with SVSolid

    f.   Specify model output

    g.   Run SVSolid model

SVSlope Steps:

    h.   Specify analysis settings

    i.   Specify dynamic programming grid

    j.   Specify search boundary coordinates

    k.   Apply SVSlope material properties

    l.   Run SVSlope model

    m. Visualize results

The details of these outlined steps are given in the following sections.

> **NOTE:**
> Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

PROFESSIONAL authorization is required for this tutorial. The steps to ensure that PROFESSIONAL authorization is activated are described in the Authorization section.

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,
2. Press the Clear Search button (if enabled),
3. Create a new project called "UserTutorial" by pressing the *New* button next to the list of projects,
4. Create a new model called "DP Example" by pressing the *New* button next to the list of models. Use the settings below when creating this new model:

        Application: SVSOLID
        Model Name: DP Example
        System: 2D
        Units: Metric

5. Click on the *World Coordinate System* tab,
6. Enter the World Coordinates System coordinates shown below into the dialog (leave Global Offsets as zero),

        x min = -5   x max = 90
        y min = -5   y max = 30

7. Click *OK* to close the dialog.

The new model will be automatically added under the recently created UserTutorial project.

SVSOLID now opens to show a grid and the Options dialog (*View > Options)* pops up. Change the default horizontal and vertical grid spacing to 1.0 m and click *OK*.

b. Enter Geometry (Model > Geometry)

Model geometry is defined as a set of regions. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from either .DXF files or from existing models.

This model will be divided into three regions, which are named Upper Layer, Weak Layer, and Lower Layer. Each region will have one of the materials listed above specified as its material properties. The shapes that define each material region will now be created.

- **Define Upper Layer Region**

    1. Select *Draw > Geometry > Polygon Region* from the menu,
    2. The cursor will now be changed to a cross hair,
    3. Move the cursor near (0,27) in the drawing space. You can view the coordinates of the current position of the mouse in the status bar,
    4. To select the point as part of the shape left click on the point,
    5. Now move the cursor near (0,13) and left-click the mouse. A line is now drawn from (0,27) to (0,13),
    6. In the same manner then enter the following points:
        (80,13)
        (80,15)
        (44,15)
    7. Move the cursor near the point (20,27). Double click on the point to finish the shape. A line is now drawn from (44,15) to (20,27) and the shape is automatically finished by SVSLOPE by drawing a line from (20,27) back to the start point, (0,27).

Repeat this process to define the Weak Layer and Lower Layer regions according to the data provided in the tables below.

Region: Upper Layer

X (m)	Y (m)
0	27
0	13
80	13
80	15
44	15
20	27

Region: Weak Layer

X (m)	Y (m)
0	13
0	12
80	12

80	13

Region: Lower Layer

X (m)	Y (m)
0	12
0	0
80	0
80	12

NOTE:
 If an error is made when entering the region geometry the user may recover from the error and start again by one of the following methods:
   a. Press the escape (esc) key.
   b. Select a region shape and press the delete key.
   c. Use the Undo function on the Edit menu.

If all model geometry has been entered correctly the shape should look like the diagram at the start of this tutorial.

- **Specify Region Names**

  1. Select *Model > Geometry > Regions* from the menu,
  2. The Regions dialog will be opened,
  3. Select the name "R1" in the list,
  4. Enter the name "Upper Layer",
  5. Select the name "R2",
  6. Enter the name "Weak Layer",
  7. Select the name "R3",
  8. Enter the name "Lower Layer",
  9. Press *OK* to close the dialog.

c. Specify Boundary Conditions (Model > Boundaries)

Boundary conditions must be applied to region points. Once a boundary condition is applied to a boundary point the starting point is defined for that particular boundary condition. The boundary condition will then extend over subsequent line segments around the edge of the region in the direction in which the region shape was originally entered. Boundary conditions remain in effect around a shape until re-defined. The user may not define two different boundary conditions over the same line segment.

More information on boundary conditions can be found in *Menu System > Model Menu > Boundary Conditions > 2D Boundary Conditions* in your User's Manual.

Now that all of the regions and the model geometry have been successfully defined, the next step is to specify the boundary conditions. The sides of the model will be fixed in the x direction to prevent lateral movement and the model will be fixed in both directions at its base. The steps for specifying the boundary conditions are as follows:

- **Upper Layer**

    1. Select the "Upper Layer" region in the region selector,
    2. From the menu select *Model > Boundaries > Boundary Conditions*. The *boundary conditions* dialog will open. By default the first boundary segment will be given a Free condition in both the x and y directions,
    3. Select the point (0,27) from the list on the Segment Boundary Conditions tab,
    4. From the X Boundary Condition drop-down select a Fixed boundary condition,
    5. Select the point (0,13) from the list,
    6. From the X Boundary Condition drop-down select a Free boundary condition,
    7. Select the point (80,13) from the list,
    8. From the X Boundary Condition drop-down select a Fixed boundary condition,
    9. Select the point (80,15) from the list,
    10. From the X Boundary Condition drop-down select a Free boundary condition,
    11. Click OK to save the input Boundary Conditions and return to the workspace,

> **N O T E :**
> The Free Y boundary condition for the point (0,27) becomes the boundary condition for the following line segments that have a Y Continue boundary condition and the Free X boundary condition for the point (0,13) becomes the boundary condition for the following line segments that have a X Continue boundary condition, until a new boundary condition is specified.

- **Weak Layer**

    1. Select the "Weak Layer" region in the region selector,
    2. From the menu select *Model > Boundaries > Boundary Conditions* to open the *Boundaries* dialog,
    3. Select the point (0,13) from the list,
    4. From the X Boundary Condition drop-down select a Fixed boundary condition,
    5. Select the point (0,12) from the list,
    6. From the X Boundary Condition drop-down select a Free boundary condition,
    7. Select the point (80,12) from the list,
    8. From the X Boundary Condition drop-down select a Fixed boundary condition,
    9. Select the point (80,13) from the list,
    10. From the X Boundary Condition drop-down select a Free boundary condition,
    11. Click *OK* to save the input Boundary Conditions and return to the workspace,

- **Lower Layer**

    12. Select the "Lower Layer" region in the region selector,
    13. From the menu select *Model > Boundaries > Boundary Conditions* to open the *Boundaries* dialog,

14. Select the point (0,12) from the list,

15. From the X Boundary Condition drop-down select a Fixed boundary condition,

16. Select the point (0,0) from the list,

17. From the X Boundary Condition drop-down select a Fixed boundary condition,

18. From the Y Boundary Condition drop-down select a Fixed boundary condition,

19. Select the point (80,0) from the list,

20. From the X Boundary Condition drop-down select a Fixed boundary condition,

21. From the Y Boundary Condition drop-down select a Free boundary condition,

22. Select the point (80,12) from the list,

23. From the X Boundary Condition drop-down select a Free boundary condition,

24. From the Y Boundary Condition drop-down select a Free boundary condition,

25. Click *OK* to save the input Boundary Conditions and return to the workspace.

d. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the materials that will be used in the model. The material names in this tutorial match the region names. This section will provide instructions on creating the materials and entering the SVSolid material parameters. The SVSlope material parameter specification will be described below.

It should be noted that the Mohr-Coulomb soil properties are required for the SVSlope portion of the analysis only. The SVSolid portion of the analysis is a linear elastic analysis and the Mohr-Coulomb properties are not required for the stress portion of the analysis. Current research into the Dynamic Programming method has shown that the difference in the computed FOS between whether an elasto-plastic strength model is used or a linear elastic strength model is used in the base finite element analysis makes for negligible difference if the FOS is greater than 1.0. Therefore a linear elastic stress analysis is more than adequate for most situations.

The steps below are instructions for the Upper Layer material. Repeat the process to add the other materials.

1. Open the *Materials* dialog by selecting *Model > Materials > Manager...* from the menu,

2. Click the *New...* button to create a material,

3. Enter "Upper Layer" for the material name in the dialog that appears and choose Linear Elastic for the Data type of this material,

4. Press *OK* to close the dialog. The *Material Properties* dialog will open automatically,

> **N O T E :**
> When a new material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned to it will display that material's fill color.

5. On the *Parameters* tab enter the SVSolid parameter values provided in the table below,

6. Check the "Apply Vertical Body Load" box,

7. Click the *OK* button to close the dialog,

8. Repeat these steps to create the Weak Layer and Lower Layer materials,

9. Press *OK* to close the *Materials Manager* dialog.

Material	SVSolid Type	γ (kN/m^3)	E (kPa)	v
Upper Soil	Linear Elastic	15	15000	0.33
Weak Layer	Linear Elastic	18	2000	0.45
Lower Soil	Linear Elastic	20	100000	0.35

Once all material properties have been entered, we must apply the materials to the corresponding regions.

1. Open the *Region Properties* dialog by selecting *Model > Geometry > Region Properties* from the menu,

2. Select the Upper Layer region and assign the *Upper Layer* material to this region,

3. Select the Weak Layer region and assign the *Weak Layer* material to this region,

4. Select the Lower Layer region and assign the *Lower Layer* material to this region,

5. Press the *OK* button to accept the changes and close the dialog.

e. Combine SVSlope with SVSolid (File > Add Coupling)

Modeling of SVSolid and SVSlope can be done independently or in "Combination" by specifying SVSolid and SVSlope components in the same model file. This methodology makes it easy to use the finite-element stress results when using the dynamic programming search method.

1. Press *File > Save* to save a copy of the steps so far in the current model, as the Add Coupling operation creates a new model,

2. Select *File > Add Coupling*,

3. The Add Coupling dialog will be displayed,

4. Check the *SVSlope* box,

5. Note that this process creates a new model file with the combined components in the same Project,

6.   Click *OK* to close the dialog.

f. Specify Model Output

PLOT MANAGER (Model > Reporting > Plot Manager)
The plot manager dialog is used to specify information to display in the solver. There are many plot types that can be specified to visualize the results of the model. The defaults will be generated for this tutorial example model.

1. Open the *Plot Manager* dialog by selecting *Model > Reporting > Plot Manager* from the menu,

2. The default plots for the model are automatically generated and displayed in the plot manager,

3. Use the Properties button to view more details on any of the plots listed,

4. Click *OK* to close the *Plot Manager* and return to the workspace.

OUTPUT MANAGER **(Model > Reporting > Output Manager)**

The output manager dialog is used to specify information to export to other software, including SVSlope and the AcuMesh visualization software.

Since a combined SVSolid/SVSlope model is being created an output file of the finite-element stress results is specified by default, in addition to the AcuMeshInput.dat file.

g. Run Model   (Solve > Analyze)

The next step is to analyze the SVSolid component of the model. Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model. After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the FlexPDE solver. Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots. This section will give a brief analysis for each plot that was generated.

These reports are intended to provide the user with low-quality graphs which give a rough indication of the results. Creating professional-quality visualizations of the results can be accomplished with ACUMESH software.

h. Specify Analysis Settings   (Model > Settings)

The Analysis Settings are set by default when the combination was performed:

1. Select *Window > SVSlope* to switch to the SVSlope environment,

2. Select *Model > Settings* from the menu,

3. Move to the *Slip Surface* tab and notice that the following items are selected:

   Slip Direction: *Left to Right*
   Slip Shape: *Non-Circular*
   Search Method: *Dynamic Programming*

4. Select the *Calculation Methods* tab to see the method type as shown below is selected:

   SAFE-DP

5. Press *OK* to close the dialog.

i. Specify Dynamic Programming Grid   (Model > Slip Surface > Dynamic Programming > Grid Points)

The stress field generated by SVSolid is used as the initial conditions for SVSlope. In a

combined SVSolid/SVSlope model the stress input file is already specified on the Initial Conditions dialog. The dynamic programming grid lines will be adjusted for this tutorial:

1.   Select *Model > Slip Surface > Dynamic Programming > Grid Points...*,

2.   Enter 31 for the *X* grid lines,

3.   Enter 121 for the *Y* grid lines,

4.   Press *OK* to close the dialog.

j.   Specify Search Boundary Coordinates (Model > Search Boundary)

The Dynamic Programming method of searching for the critical slip surface has already been selected. Now the search boundary must be defined. This is accomplished through the following steps:

1.   Select *Model > Slip Surface > Dynamic Programming > Search Boundary...*,

2.   The *Search Boundary* dialog will open with the default search boundary coordinates encompassing most of the model,

3.   Refer to the list of search boundary coordinates provided in the table below and enter the coordinates in the appropriate text boxes,

4.   Close the dialog by clicking *OK*.

The adjusted search boundary graphics are now be displayed on the CAD window.

Point	Value (m)
Top Y	31.050
Int Y	19.800
Bottom Y	5.850
Left X	8.000
Int 1 X	21.333
Var X	34.667
Int 2 X	45.333
Right X	64
VarY	17.775

k.   Apply SVSlope Material Properties (Model > Materials)

Previously, the materials for the model were defined in SVSolid and assigned to regions. Now the SVSlope properties need to adjusted for those same materials. This section will provide instructions on adjusting the Upper Layer material. Repeat the process to adjust the other materials. Refer to the beginning of this tutorial for the list of material properties. Note that the Bedrock material type does not require entry of any values for SVSlope.

1.   Open the *Materials* dialog by selecting *Model > Materials > Manager...* from the menu,

2.   Select the Upper Layer material and press *Properties* to to open the *Material*

Properties dialog,

3. On the *Shear Strength* tab enter the parameter values provided in the table below,

4. Click the *OK* button to close the *Material Properties* dialog,

5. Repeat these steps to adjust the Weak Layer material,

6. Select the Lower Layer and press the *Change Type* button,

7. Click the *OK* button,

8. Click the *OK* button to close the *Material Manager* dialog.

Material	SVSlope Type	c (kPa)	ϕ (deg)	γ (kN/m^3)
Upper Soil	Mohr Coulomb	10	30	15
Weak Layer	Mohr Coulomb	0	10	18
Lower Soil	Bedrock	-	-	-

l.   Run SVSlope Model   (Solve > Analyze)

The next step is to analyze the model.

1. Select *Solve > Analyze* from the menu. A pop-up dialog will appear and the solver will start,

2. Press the OK button to close the dialog.

m.   Visualize Results   (Window > AcuMesh)

After the model has been run, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on *Yes*. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH as appears in the diagrams following. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen.

6.2   Results and Discussions

If the model has been appropriately entered into the software the following results should be shown for the SAFE method.
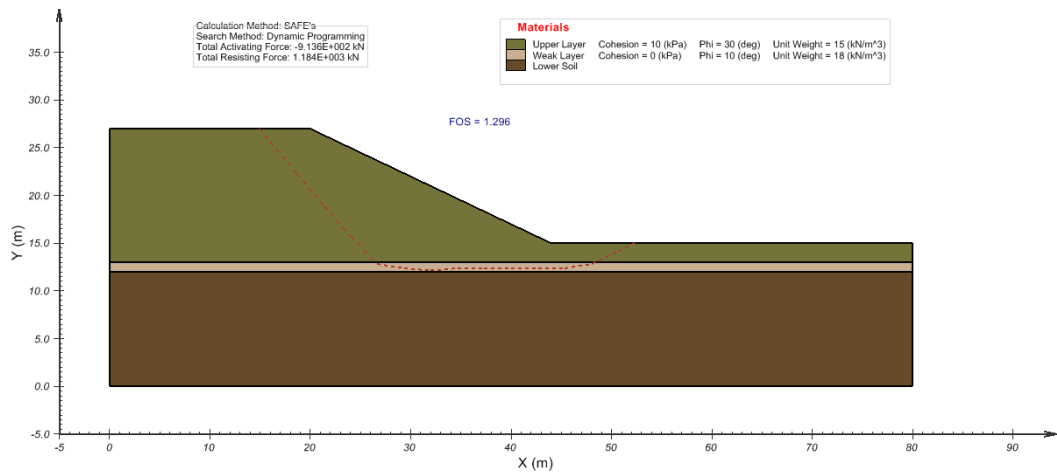
The correct results for this example are:

Method		
	SVSLOPE	
	Moment	Force
SAFE		1.296

7   Kulhawy Method

The dynamic programming example presented in Section 5 is extended to incorporate the Kulhawy stress-based slope stability analysis method.

This original model can be found under:

Project: Slopes_STUDENT
Model: SAFE_52_Kulhawy_EDU

Minimum authorization required to complete this tutorial: PROFESSIONAL

Model Description and Geometry
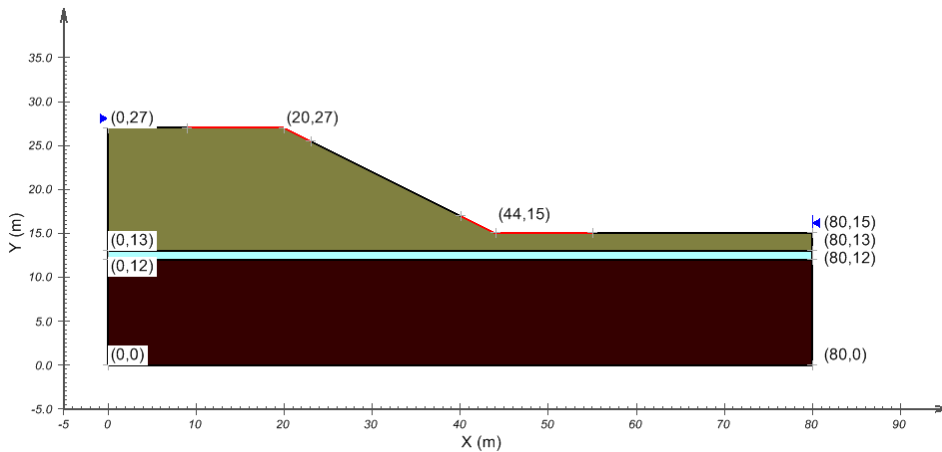
Entry and Exit

Entry Range				Exit Range		
Left Side				**Right Side**		
	X	Y			X	Y
Left Point	9	27		Left Point	40	17
Right Point	23	25.5		Right Point	55	15
Increments		12		Increments		12

7.1   Model Setup

The following steps will be required to set up this model:

  a.   Create model
  b.   Specify analysis settings
  c.   Specify search method geometry

    d.   Run SVSolid model

    e.   Run SVSlope model

    f.   Visualize results

> **N O T E :**
> Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

Since PROFESSIONAL authorization is required for this tutorial, perform the following steps to ensure PROFESSIONAL authorization is activated:

1. Plug in the USB security key,

2. Go to the *File > Authorization* dialog on the SVOFFICE Manager,

3. Software should display PROFESSIONAL authorization. If not, it means that the security codes provided by SoilVision Systems at the time of purchase have not yet been entered. Please see the Authorization section of the SVOFFICE User's Manual for instructions on entering these codes.

In order to create the Kulhawy model, save a copy of the Dynamic Programming Example model. This is accomplished through the following steps:

1. Select the "UserTutorial" project and Open the "DP Example" model,

2. Select *File > Save As*,

3. Type the name User_Kulhawy and click *OK*.

Now a new model has been created and loaded into the workspace that will be modified to include Kulhawy analysis.

b. Specify Analysis Settings (Model > Settings)

In SVSlope the Analysis Settings provide the information for what type of analysis will be performed. These settings will be specified as follows:

1. Enter SVSlope mode, if not already (*Window > SVSlope* from the menu)

2. Select *Model > Settings* from the menu,

3. Select the Slip Surface tab,
    Slip Direction: Left to Right
    Slip Shape: Composite Circular
    Search Method: Entry and Exit

4. Select the *Calculation Methods* tab from the dialog and select the method types as shown below:
    Spencer
    Morgenstern-Price
    GLE (Fredlund)
    Kulhawy

5. Press *OK* to close the dialog.

c. Specify Search Method Geometry

The Entry and Exit method of searching for the critical slip surface has already been selected in the previous step. Now the user must draw the graphical representation of the entry range and exit range on the screen. This is accomplished through the following steps:

1. Open the *Entry and Exit* dialog through the *Model > Slip Surface > Entry and Exit...* menu option,

2. Enter the values for the entry range and exit range as specified at the start of this tutorial (the range values may also be drawn on the CAD window),

3. Click *OK* to close the dialog.

d. Run SVSolid Model (Solve > Analyze)

The next step is to analyze the SVSolid component of the model. Select Win*dow > SVSolid* from the menu. Then Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver for the SVSolid component of the model. The solver will automatically begin solving the model. After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the FlexPDE solver. Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots.

These reports are intended to provide the user with low-quality graphs which give a rough indication of the results. Creating professional-quality visualizations of the results can be accomplished with ACUMESH software.

e. Run SVSlope Model (Solve > Analyze)

The next step is to analyze the SVSlope component of the model.

1. Select Win*dow > SVSlope* from the SVOffice menu.

2.   Select *Solve > Analyze* from the menu. A pop-up dialog will appear and the solver will start,

3. Press the OK button to close the dialog.

f. Visualize Results (Window > AcuMesh)

After the model has been run, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click the *Acumesh...* button. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH as appears in the diagrams following. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen.

7.2 Results and Discussions

If the model has been appropriately entered into the software the following results should be shown. The results will contain all trial slip surfaces as well as the most critical slip surface results. In order to identify the most critical slip surface the user may perform the following steps:

1. Select "Slip Surfaces" from the menu item Slips, and

2. Click the *Show Trial Slip Surfaces* button, this will cause all the trial slip surfaces

to not be displayed.

The user may also plot the slices used in the analysis of the critical slip surfaces through the slips show slices menu option. The information on any particular slice may be displayed through the slips slice information dialog. A slice information dialog will appear and the user may click on a new particular slice on the slope to display the details of that slice. The user may display results from different methods by clicking the combo box on the display which lists the different analysis methods (Spencer, Morgentern-Price, etc.). The analysis results in a factor of safety of 1.393 for the Kulhawy method.
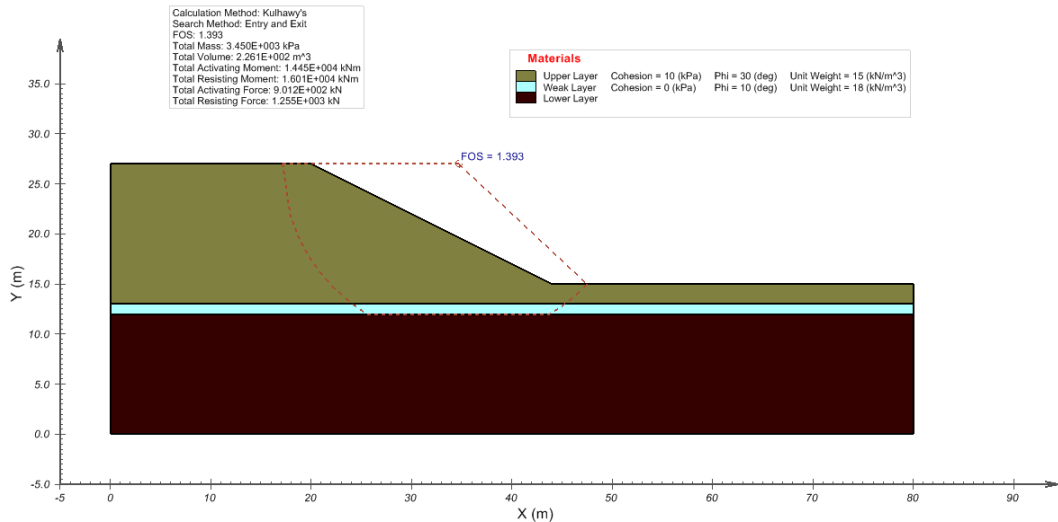
8   2D Hong Kong Example

This example is used to illustrate the use of a coupled seepage and slope stability model considering the influence of climatic rainfall events on the resulting factor of safety. The climatic effects are considered over a period of three days. The combination of SVFlux and SVSolid and are used to model a transient change in pore-water pressure and the resultant changes in the factor of safety for the slope. The model is based on site data taken from a slope in Hong Kong.

This example consists of a four-layer slope with a rainfall event applied to top of the slope. The purpose of this model is to illustrate the effect of the infiltration of the rainfall into the soil on the factor of safety for the slope.

This original model can be found under:

Project: Slopes_Group_3
Model: HongKongExample1_Unsaturated_Rain

Minimum authorization required to complete this tutorial: PROFESSIONAL
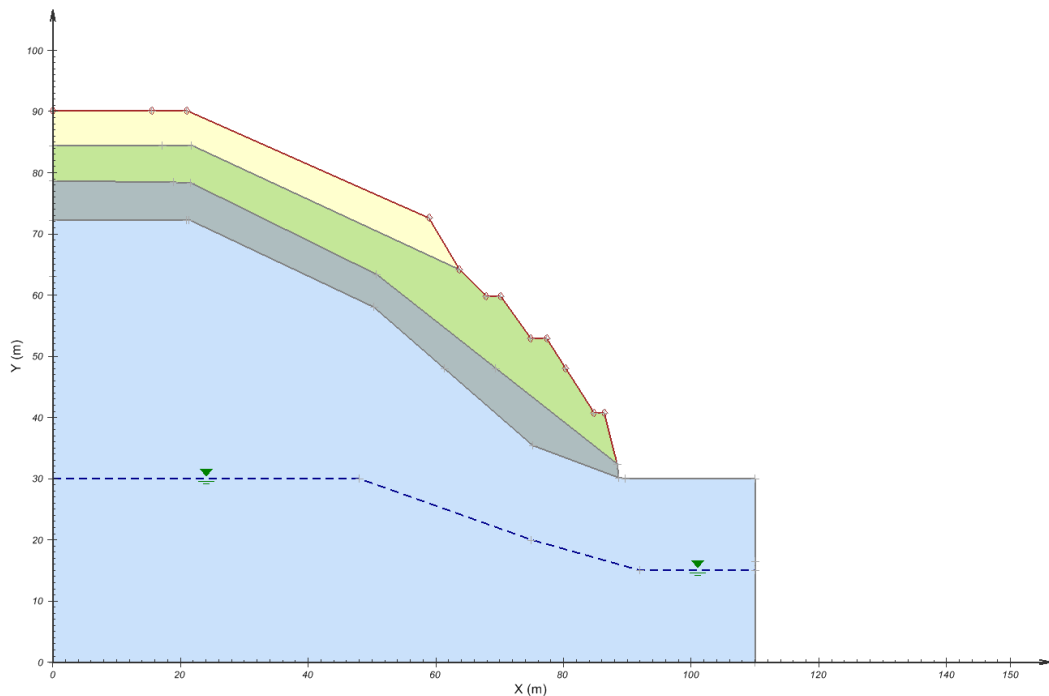
Model Geometry

8.1   Model Setup

The following steps will be required to set up the model:

SVFlux steps:

- a.   Create model
- b.   Enter geometry
- c.   Specify SVFlux initial conditions
- d.   Specify SVFlux boundary conditions
- e.   Apply SVFlux material properties
- f.   Combine SVSlope with SVFlux
- g.   Specify SVFlux model output
- h.   Run SVFlux model

SVSlope steps:

- i.   Specify analysis settings
- j.   Specify search method geometry
- k.   Specify pore-water pressure
- l.   Apply SVSlope material properties
- m.   Run SVSlope model
- n.   Visualize results

The details of these outlined steps are detailed in the following sections.

> **N O T E :**
> Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a.   Create Model

PROFESSIONAL authorization is required for this tutorial. The steps to ensure that PROFESSIONAL authorization is activated are described in the Authorization section.

This model is first created as a seepage only model in SVFlux. Later in the tutorial the slope stability product SVSlope will be coupled with SVFlux.

To begin this tutorial create a new model in SVFLUX through the following steps:

- 1.   Select a Project under which to organize the tutorial,
- 2.   Press the SVFlux icon found above the list of models,
- 3.   Enter "Hong Kong Example" in the Model Name box,
- 4.   Select the following entries:

    Application: SVFLUX

System:	2D
Type:	Transient
Units:	Metric
Time Units:	day

5. Click on the *World Coordinate System* tab,

6. Enter the World Coordinates System coordinates shown below into the dialog (leave global offsets as zero),

    x min = 0   x max = 150

    y min = 0   y max = 100

7. Click on the *Time* tab,

8. Enter the following values for time:

    Start Time: 0

    Initial Increment: 0.1

    Maximum Increment: 0.2

    End Time: 3

9. Click the *OK* button to save the model and close the *New Model* dialog,

10. The *Options* dialog will appear. Click *OK* to accept a default horizontal and vertical spacing of 1m.

b. Enter Geometry (Model > Geometry)

Model geometry is defined as a set of regions and a series of layers. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from either .DXF files or from existing models.

This model will be divided into four regions, which are named R1, R2, R3, and R4. The shapes that define each material region can be created by the following steps.

1. Open the Regions dialog by selecting Model > Geometry > Regions... from the menu,

2. Click the *New* button 3 times to create the second, third and fourth regions,

3. Select the region R1 and click the *Properties...* button to open the *Region Properties* dialog,

4. Click the *New Polygon...* button to open the *New Polygon Shape* dialog,

5. Copy the region coordinate data (do not copy the X and Y header row) for R1 provided at the end of this tutorial and click the *Paste* button on the *New Polygon Shape* dialog to paste the region data into the data grid,

6. Click *OK* to close the dialog and create the new region,

7. Click the right arrow at the top right of the Region Properties dialog to move to the second region R2,

8. Repeat the steps preformed for R1 to create regions R2, R3, and R4,

9. Click OK on the *Region Properties* dialog and on the *Regions* dialog to accept the region changes.

If all model geometry has been entered correctly the shape should look like the diagram at the beginning of this tutorial.

c. Specify SVFlux Initial Conditions (Model > Initial Conditions)

Initial conditions must be specified prior to solving a transient seepage model. In this case we will specify a water table as an initial condition.

1. Select *Model > Initial Conditions > Settings... from the menu,*
2. Select the "Water Table" option and click *OK* to close the dialog,
3. Select Model > Initial Conditions > Water Table... from the menu,
4. Either copy and paste the water table data from the tale below into the data grid on the dialog using the *Paste Points* button or enter the coordinates into the data grid manually,
5. Click OK to close the *Initial Water Table* dialog,

X (m)	Y (m)
0	30
48	30
75	20
92	15
110	15

d. Specify SVFlux Boundary Conditions (Model > Boundaries)

Now that all of the regions and the model geometry have been successfully defined, the next step is to specify the boundary conditions. A climate boundary condition will be applied to the overburden ground surface.

A climate boundary condition will be applied to the model to simulate rainfall. The steps for specifying the boundary conditions are as follows:

1. Open the *Climate Manager* dialog by selecting *Model > Boundaries > Climate Manager...* from the menu,
2. Click the *New* button to open the *New Climate Data* dialog,
3. Enter "Rainfall" as the climate dataset name,

- **Define Precipitation**

4. Double-click the precipitation cell for the "Rainfall" entry to open the *Precipitation Properties* dialog,
5. Check "Include",
6. For Input Option select "Data - Global Intensity",
7. Set the Intensity Type to "Parabolic",

8. Enter the data provided below in the data table. The data can be cut and pasted from the table below into the dialog,

9. Switch to the *Global Intensity* tab,

10. Enter 8 hrs for the Intensity Start,

11. Enter 14 hrs for the Intensity End,

12. Click *OK* to save and close the *Precipitation Properties* dialog.

Time (day)	Flux (m³/day/m²)
0	0.05
1	0
2	0

- **Apply boundary conditions to geometry**

To apply the Rainfall precipitation event to the model ground surface perform the following steps:

1. Select the top region by clicking on the region (or select R1 from the region selector),

2. From the menu select *Model > Boundaries > Boundary Conditions...*. The *Boundary Conditions* dialog will open. By default the first boundary segment is given a "No BC" value,

3. Select the point (0, 90.115) in the boundary conditions list,

4. From the Boundary Condition drop down select a "Climate" boundary condition,

5. Select "Rainfall" as the name of the climate object to apply,

6. From the Boundary Condition drop down select a "Continue" boundary condition for the next 3 points in the boundary conditions list: (15.554, 90.115), (21.047, 90.115), and (58.915, 72.586),

7. For the point (63.723, 64.147), select a "No BC" boundary condition from the Boundary Condition drop down,

8. Click *OK* to any pop-ups that appear,

9. Click *OK to close the Boundary Conditions dialog.*

10. Select the second region by clicking on the region (or select R2 from the region selector),

11. From the menu select *Model > Boundaries > Boundary Conditions...*. The *Boundary Conditions* dialog will open. By default the first boundary segment is given a "No BC" value,

12. Select the point (63.723, 64.127) in the boundary conditions list,

13. From the Boundary Condition drop down select a "Climate" boundary condition,

14. Select "Rainfall" as the name of the climate object to apply,

15. From the Boundary Condition drop down select a "Continue" boundary condition

for the next 7 consecutive points in the boundary conditions list,

16. For the next point in the list (88.396, 32.423), select a "No BC" boundary condition from the Boundary Condition drop down,

17. Click *OK* to any pop-ups that appear,

18. Click *OK to close the Boundary Conditions dialog.*

e. Apply SVFlux Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the four SVFlux materials used in the model. The parameter values for all materials are provided in the table below. The SWCC and hydraulic conductivity data for the Colluvium and WeatheredGranite materials is provided at the end of this tutorial. Note that the LessWeatheredGranite and Bedrock materials do not contain SWCC and hydraulic conductivity data so the Fredlund and Xing fitting parameters af, nf, mf, and hr must be entered directly into the *Fredlund & Xing Fit* dialog by the user rather than using the *Apply Fit* button to have the software calculate these values.

1. Open the *Materials* dialog by selecting *Model > Materials > Manager...* from the menu,

2. Click the *New...* button to open the *New Materials* dialog,

3. Enter "Colluvium" for the material name,

4. Set Data Type to Unsaturated,

5. Click *OK* and the *Material Properties* dialog will open,

6. Enter the parameter values and data for the Colluvium material as provided in the table below and at the end of this tutorial,

7. Click *OK* to save and close the *Material Properties* dialog,

8. Repeat the above steps to create the remaining three materials,

9. Press *OK* on the *Materials Manager* dialog to close this dialog.

Parameter	Material			
	Colluvium	WeatheredGranite	LessWeatheredZone	Bedrock
Saturated VWC	0.41	0.40	0.40	0.40
SWCC Method	Fredlund & Xing	Fredlund & Xing	Fredlund & Xing	Fredlund & Xing
Fredlund & Xing af (kPa)	7.57	4.30	50	150
Fredlund & Xing nf	0.99	8.89	0.7	0.7
Fredlund & Xing mf	0.65	0.19	0.5	0.5
Fredlund & Xing hr (kPa)	219.46	11.26	3000	3000
ksat (m/day)	2.59	0.61	0.52	0.43
Unsaturated Hydraulic Conductivity Method	Modified Campbell	Modified Campbell	Modified Campbell	None
Modified Campbell p	5	5	5	N/A
k-minimum (m/day)	8.64E-06	8.64E-06	8.64E-06	8.64E-06

- **Assign materials to regions**

The next step is to define which materials are applied to which regions.

1. Select *Model > Geometry > Regions...*,
2. For each region the appropriate material type must be selected from the combo box. The material assignments should be as follows:

    R1:   Colluvium

    R2:   Weathered Granite

    R3:   LessWeatheredGranite

    R4:   Bedrock

3. Click *OK* once the material assignments have been made.

f. Combine SVSlope with SVFlux   (File > Add Coupling)

Modeling of SVFlux and SVSlope can be done independently or in "Combination" by specifying SVFlux and SVSlope components in the same model file. This methodology makes it easy to use the finite-element pore-water pressure results in the slope stability software. The steps to combine SVSlope with SVFlux are as follows:

1. Press *File > Save* to save a copy of the steps so far in the current model, as the Add Coupling operation creates a new model,
2. Select *File > Add Coupling...*,
3. The *Add Coupling* dialog will be displayed,
4. Check the *SVSlope* box,
5. Note that this process creates a new model file with the combined components in the same Project,
6. Enter "Hong Kong Example coupled" as the New File Name,
7. Click *OK* to close the dialog.

The remaining steps in this tutorial are related to the SVSlope part of the model. To switch the model view to the SVSlope component of the model select *Window > SVSlope* from the menu or click on the SVSlope icon on the sidebar near the top left of the screen.

g. Specify SVFlux Model Output

OUTPUT MANAGER   (Model > Reporting > Output Manager)

The output manager dialog is used to specify information to export to other software, including SVSlope and the AcuMesh visualization software.

Since a combined SVFlux/SVSlope model is being created an output file of the finite-element pore-water pressure results is specified by default, in addition to the default

AcuMeshInput.dat.

h. Run Model (Solve > Analyze)

The next step is to analyze the model. Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model. After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the FlexPDE solver. Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots. These reports are intended to provide the user with low-quality graphs which give a rough indication of the results. Creating professional-quality visualizations of the results can be accomplished with ACUMESH software.

i. Specify Analysis Settings (Model > Settings)

In SVSlope the Analysis Settings provide the information for what type of analysis will be performed. These settings are specified as follows:

1. Select *Model > Settings...* from the menu,

2. Select the Slip Surface tab,
   Slip Direction: Left to Right
   Slip Shape: Circular
   Search Method: Entry and Exit

3. Select the *Calculation Methods* tab from the dialog and select the method types as shown below:
   Spencer
   Morgenstern-Price
   GLE (Fredlund)

4. Press *OK* to close the dialog.

j. Specify Search Method Geometry

The Entry and Exit method of searching for the critical slip surface has already been selected in the previous step. Now the user must specify the geometry that defines the search method. This is accomplished through the following steps:

1. Open the *Entry and Exit* dialog through the *Model > Slip Surface > Entry and Exit...* menu option,

2. Enter the values for the entry range and exit range as specified in the table below,

3. Click *OK* to close the dialog.

Entry Range				Exit Range		
	Left Side				**Right Side**	
	Left Point	**Right Point**			**Left Point**	**Right Point**
X	17	41.686		X	78.269	90
Y	90.115	80.561		Y	51.421	30

Increments	6	Increments	6

k. Specify Pore-Water Pressure (Model > Pore Water Pressure)

Pore-water pressure profiles from the SVFlux solution are used in the SVSlope model. To specify which profiles are to be used in solving the SVSlope model follow these steps:

1. Select *Model > Pore Water Pressure > Settings...*,
2. Select the "PWP Times" tab to view the available pore-water pressure profiles,
3. Ensure that all times are checked,
4. Press *OK* to close the dialog and accept this list of time values.

l. Apply SVSlope Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the four materials that will be used in the SVSlope model. This section will provide instructions on creating the Colluvium material. Repeat the process to add the other materials.

1. Open the *Materials* dialog by selecting *Model > Materials > Manager...* from the menu,
2. Click the *New* button to create a material,
3. Enter "Colluvium" for the material name in the dialog that appears and choose *Unsaturated Phi-b* for the Shear Strength type of this material,
4. Press *OK* to close the dialog. The *Material Properties* dialog will open automatically,
5. Enter the parameter values provided in the table below for each of the four materials,
6. Click the *OK* button to close the *Material Properties* dialog,

Material	Shear Strength Type	Cohesion (kPa)	Friction Angle phi (deg)	Friction Angle phi-b (deg)	Unit Weight (kN/m³)
Colluvium	Unsaturated Phi-b	10	35	10	19.6
Weathered Granite	Unsaturated Phi-b	15.1	35.2	10	19.6
LessWeatheredGranite	Unsaturated Phi-b	23.5	41.5	10	19.6
Bedrock	Mohr Coulomb	50	40	N/A	25

The material properties have already been assigned to regions in the SVFlux component of this model so they do not need to be re-assigned in SVSlope.

m. Run SVSlope Model (Solve > Analyze)

The next step is to analyze the SVSlope component of the model.

1. Select *Solve > Analyze* from the menu. A pop-up dialog will appear and the solver will start.

n. **Visualize Results** (Window > AcuMesh)

After the model has completed solving, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on the *Acumesh...* button. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen.

8.2   Results and Discussion

The results for the calculation of the factor of safety may be seen below. The user may display results from different calculation methods by using the combo box on the display which lists the different analysis methods (Spencer, M-P, etc.). By default, the critical slip surface is displayed for the selected time in the combo box at the top of the workspace. All trial slip surfaces may be displayed by following these steps:

1. Select *Slips > Slip Surfaces...* from the menu,
2. Click the *Show Trial Slip Surfaces* check box.

The analysis results in a factor of safety of 1.605 for the Spencer method at time 0 days and 1.564 at time 2 days as shown in the following screenshots.

Calculation Method: Spencer's
Search Method: Entry and Exit
FOS: 1.564
Time: 3 day
Total Weight: 7.763E+003 kN
Total Volume: 3.961E+002 m^3
Total Activating Moment: 5.547E+005 kNm
Total Resisting Moment: 8.672E+005 kNm
Total Activating Force: 3.705E+003 kN
Total Resisting Force: 5.822E+003 kN

FOS = 1.564

FOS
0
0.522
1.043
1.565
2.087
2.609
3.13
3.652
4.174
4.696
5.217
5.739
6.261+
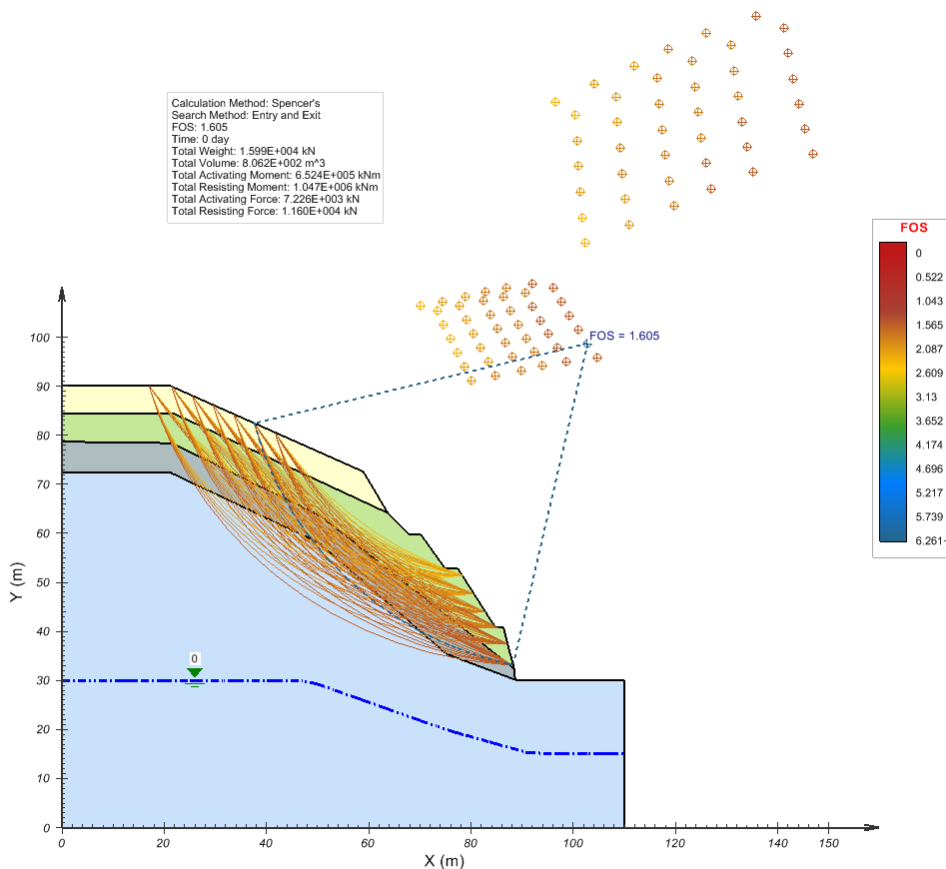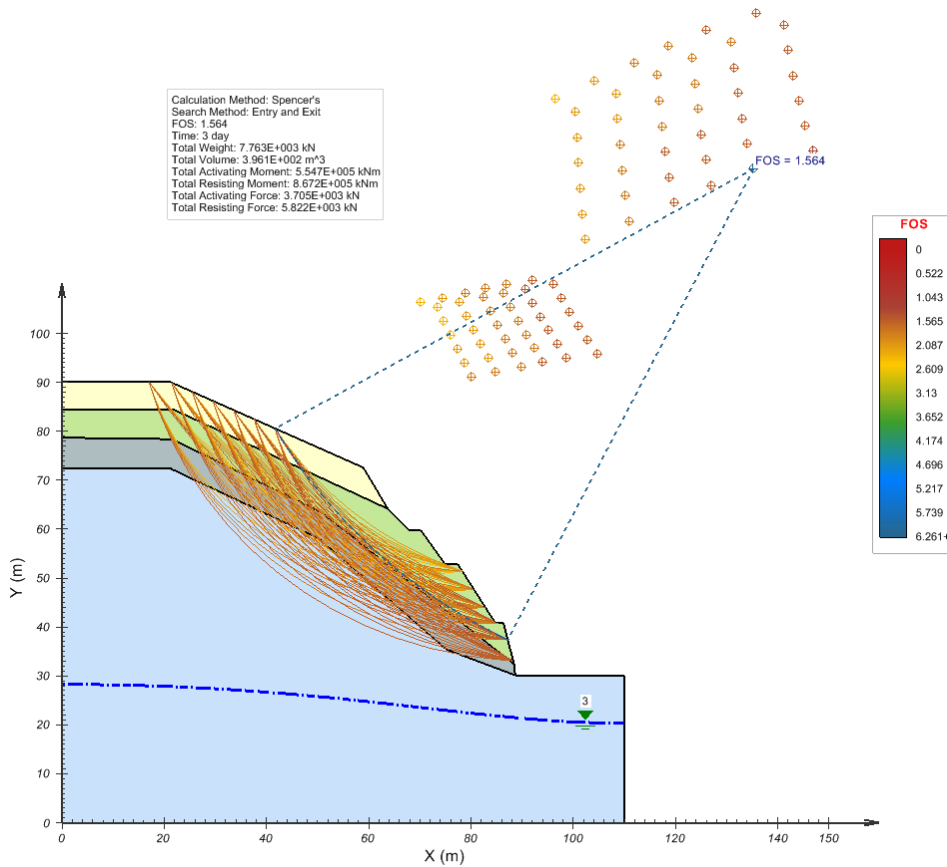
The factor of safety vs time may be viewed as follows. Select *Slips > FOS vs. Time...* from the menu. The graph shows the factor of safety decreasing over time. The rainfall event ends at 1 day but the factor of safety continues to decrease until about 2.5 days as the rain infiltrates the soil. After 2.5 days the factor of safety begins to rise as the soil begins to de-saturate.
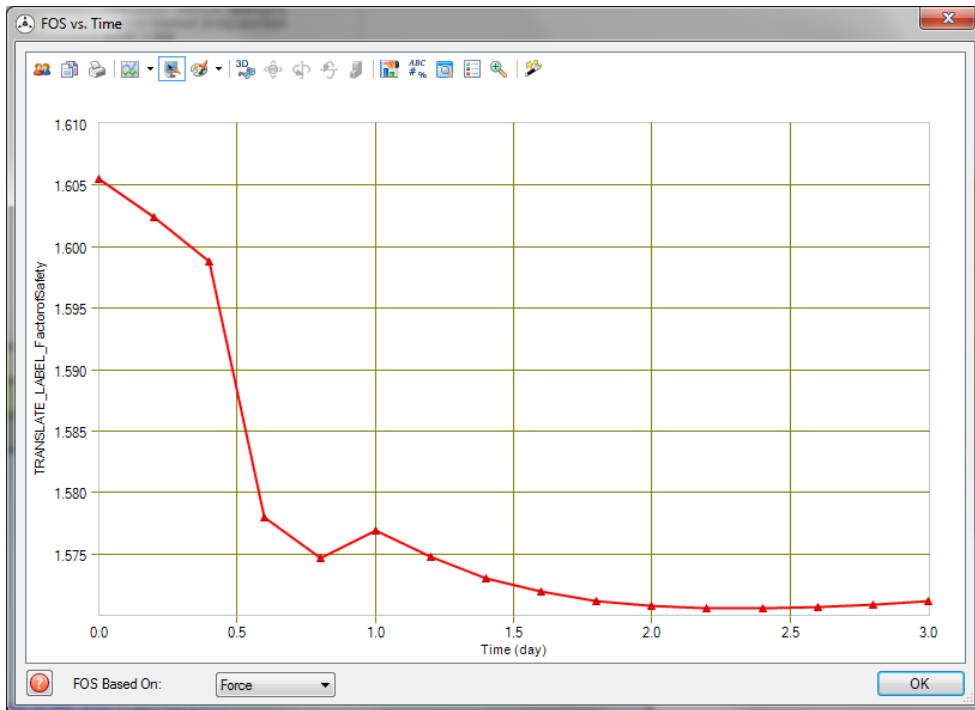
8.3   Model Data

Region Geometries

Region: R1

X (m)	Y (m)
0	90.115
15.554	90.115
21.047	90.115
58.915	72.586
63.723	64.147
21.811	84.443
17.222	84.443
0	84.443

Region: R2

X (m)	Y (m)
0	84.443
17.222	84.443
21.811	84.443
63.723	64.147

67.905	59.81
70.162	59.795
74.881	52.917
77.364	52.9
80.362	48
84.775	40.787
86.355	40.776
88.396	32.423
69.444	48.0
50.78	63.341
21.567	78.382
18.995	78.418
0	78.68

Region: R3

X (m)	Y (m)
0	78.68
18.995	78.418
21.567	78.382
50.78	63.341
69.444	48
88.396	32.423
88.654	30.151
75.193	35.403
61.328	48
50.282	58.036
21.324	72.32
20.993	72.32
0	72.32

Region: R4

X (m)	Y (m)
0	72.32
20.992	72.32
21.324	72.32
50.282	58.036
61.328	48
75.193	35.403
88.654	30.151

89.712	30
110	30
110	16.558
110	15
110	0
0	0
0	30
0	63.149

Return to Model Geometry Section

SVFlux Material Properties

Colluvium:

SWCC data:

Suction (kPa)	VWC
0.37	0.41
2.20	0.39
3.41	0.37
5.18	0.35
8.07	0.34
10.95	0.33
15.52	0.31
20.09	0.30
26.34	0.28
32.60	0.26
42.21	0.24
55.17	0.23
77.67	0.21
121.96	0.20
179.71	0.18
238.58	0.17

Hydraulic conductivity data:

Suction (kPa)	k (m/day)
0.91	2.50
2.25	2.50
3.35	2.45

4.49	2.40
5.65	2.34
6.69	2.13
7.59	1.94
8.62	1.70
9.79	1.50
10.89	1.31
21.28	0.28
31.51	0.09
41.87	0.04
52.10	0.02
63.37	0.01
73.81	7.64E-3
85.97	4.97E-3
95.95	3.45E-3
104.83	2.48E-3
155.31	7.80E-3
206.47	3.19E-3
262.48	1.60E-4
306.36	9.09E-5
356.85	5.92E-5
407.08	3.85E-5
463.68	2.77E-5
506.57	1.99E-5
552.87	1.53E-5
603.40	1.17E-5

Weathered Granite:

SWCC data:

Suction (kPa)	VWC
0.09	0.40
1.81	0.39
3.57	0.38
4.18	0.37
4.80	0.36
4.88	0.34
5.50	0.33
6.13	0.32
6.76	0.30

7.38	0.29
8.01	0.28
8.61	0.27
9.24	0.26
9.83	0.25
11.00	0.24
13.84	0.23
15.57	0.22
17.86	0.21
23.51	0.20
33.64	0.19
61.11	0.18
128.90	0.17
264.40	0.16

Hydraulic conductivity data:

Suction (kPa)	k (m/day)
0.92	0.59
2.30	0.59
3.42	0.58
4.40	0.57
5.54	0.56
6.69	0.50
7.59	0.48
8.81	0.40
9.59	0.36
10.67	0.30
21.50	3.50E-2
31.39	7.39E-3
41.91	2.24E-3
52.32	9.46E-4
62.51	4.56E-4
73.03	2.43E-4
85.19	1.43E-4
95.27	8.69E-5
106.39	5.84E-5
152.09	1.23E-5
203.12	3.72E-6
253.40	1.62E-6
303.06	7.32E-7

346.59	4.03E-7
404.32	2.37E-7
452.17	1.44E-7
494.74	9.37E-8
540.50	6.72E-8
590.80	4.67E-8

Return to Apply SVFlux Material Properties Section

9   2D Cannon Dam Example

The Cannon Dam Model was published by Wolff and Harr (1987). The probabilistic analysis results from SVSLOPE using the Monte Carlo method are compared to the results published in the paper by Wolff and Harr for noncircular slip surfaces.

Wolff and Harr (1987) used the point-estimate method for their probability analysis failure for the Cannon Dam. The location of critical slip surface was assumed fixed and taken from their paper. The friction angle input parameter for the Phase I and Phase II fills was calculated. The unit weights of the fills were back-calculated in order to match the factor of safety computed by Wolff and Harr.

The results published by Wolff and Harr (1987) were compared to those obtained by the GLE and the Spencer methods. It is assumed in the SVSLOPE model that all the probabilistic input variables are normally distributed.

This original model can be found under:
Project: Verification_SVSlope_Group1
Model: VS_34

Minimum authorization required to complete this tutorial: FULL

Model Geometry

9.1   Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

     a.   Create model
     b.   Specify analysis settings
     c.   Enter geometry
     d.   Specify search method geometry
     e.   Specify Pore-Water Pressure

    f.   Apply material properties

    g.   Run model

    h.   Visualize results

The details of these outlined steps are detailed in the following sections.

> **N O T E :**
> Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,

2. Press the *Clear Search* button (if enabled),

3. Create a new project called *UserTutorial* by pressing the *Create New Project* button above the list of projects,

4. Create a new model called "Probabilistic Example" by pressing the SVSLOPE icon button above the list of models. Use the settings below when creating this new model:

      Module: SVSLOPE

      System: 2D

      Units: Imperial

      Slope Direction: Left to Right

5. Click on the *World Coordinate System* tab,

6. Enter the World Coordinates System coordinates shown below into the dialog,

      x min = -400   x max = 600

      y min = 350   y max = 750

7. Click on *OK*.

SVSLOPE now opens to show a grid and the Options menu pops up.

The workspace grid spacing needs to be set to aid in defining region shapes. The filter portion of the model has coordinates of a precision of 0.5m. In order to effectively draw geometry with this precision using the mouse, the grid spacing must be set to a maximum of 0.5.

1. Under the grid spacing area enter 1 for both the horizontal and vertical spacing,

2. Click *OK* to close the dialog.

This menu can be reached again by selecting *View > Options...* from the menu.

b. Specify Analysis Settings (Model > Settings)

In SVSlope the Analysis Settings provide the information for what model output will be available in Acumesh. These settings will be specified as follows:

1. Select *Model > Settings...* from the menu,

2. Move to the *Slip Surface* tab and ensure that the following items are selected:

Slope Direction:	*Left to Right*
Slip Shape:	*Non-Circular*
Search Method:	*Fully-Specified*

3. Select the *Calculation Methods* tab from the dialog and select the method types as shown below:

   Spencer

   GLE

4. Move to the *Sensitivity/Probability* tab and select the *Probabilistic Analysis* option,

5. Enter the following Probabilistic Parameters:

Sampling Method:	Monte-Carlo
Number of Samples:	15000
Generator Seed:	500

6. Select a "Fixed" critical slip surface location,

7. Press *OK* to close the dialog.

8. Move to the *PWP* tab and enter the Pore Water Pressure (as is shown at the start of this tutorial), and the *Ru/B_Bar > None* is chosen,

c. Enter Geometry (Model > Geometry)

Model geometry is defined as a series of layers and can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from either .DXF files or from existing models.

This model will be divided into seven regions, named R1 to R7. The shapes that define each region are created by the following steps.

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions...* from the menu,

2. Click the *New* button 6 times to create the necessary regions,

3. Select the region *R1* and click the *Properties...* button to open the *Region Properties* dialog,

4. Click the *New Polygon...* button to open the *New Polygon Shape* dialog,

5. Copy the region coordinate data (do not copy the X and Y header row) for *R1* provided at the end of this tutorial and click the *Paste* button on the *New Polygon Shape* dialog to paste the region data into the data grid,

6. Click *OK* to close the dialog and create the new region,

7. Click the right arrow at the top right of the Region Properties dialog to move to the second region *R2*,

8. Repeat the steps performed for *R1* to create the remaining regions,

9. Click OK on the *Region Properties* dialog and on the *Regions* dialog to accept the region changes.

If all model geometry has been entered correctly the shape should look like the diagram at the beginning of this tutorial.

d. Specify Search Method Geometry (Model > Slip Surface > Fully-Specified > Linear Segments)

This model makes use of a fully-specified search methodology. The linear segments shape is used as the slip surface geometry. The geometry is specified through the following steps:

1. Select *Model > Slip Surface > Fully-Specified > Linear Segments...* from the menu,
2. Copy the linear segment data from the table provided below (do not copy the header row) and enter the data into the dialog using the *Paste* button,
3. Click *OK* to close the dialog.

X (ft)	Y (ft)
-160.271	660.926
-47.496	545.537
-6.236	514.364
26.771	514.226
286.262	515.23
343.636	558.134

e. Specify Pore-Water Pressure (Model > Pore Water Pressure)

A water table or a piezometric line must be specified as an initial condition for this model. In this model a piezometric line will be used. In order to specify that a piezometric line will be entered the user needs to following these steps:

1. Select *Model > Pore Water Pressure > Settings...*,
2. Select "Water Surfaces" as the Pore-Water Pressure Method,
3. Press *OK* to close the dialog.

The user must then proceed to enter the piezometric line coordinates:

1. Select *Model > Pore Water Pressure > Piezometric Line...*,
2. Under the *Points* tab click on the *New Line* button and enter the *X* and *Y* coordinates as provided in the table below for the piezometric line,
3. Under the Apply to Regions section ensure the check boxes for all regions are checked by clicking the *Select All* button,
4. Press *OK* to close the dialog.

X (ft)	Y (ft)
-364.734	556.402
594.317	555.486

f. Apply Material Properties (Model > Materials > Manager)

The next step in defining the model is to enter the material properties for the six materials that will be used in the model. This section will provide instructions on creating the first material. Repeat the process to add the remaining materials.

1. Open the *Materials* dialog by selecting *Model > Materials > Manager...* from the menu,

2. Click the *New...* button to create a material,

3. Enter *Phase I Fill* for the material name and choose *Mohr Coulomb* for the material Shear Strength type,

4. Press *OK* to close the dialog. The *Material Properties* dialog will open automatically,

> **N O T E :**
>
> When a new material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned to it will display that material's fill color.

5. Enter the parameter values for the *Phase I Fill* material as provided in the table below,

6. Click the *OK* button to close the *Shear Strength* dialog,

7. Repeat these steps for the remaining materials using the values in the following table.

Material	Shear Strength Type	Unit Weight (lb/ft^3)	Cohesion (psf)	Friction Angle phi (deg)
Phase I Fill	Mohr Coulomb	150	2230	6033
Phase II Fill	Mohr Coulomb	150	2906.1	14.8
Material 3	Mohr Coulomb	150	1	50
Material 4	Mohr Coulomb	150	1	35
Spoil Fill	Mohr Coulomb	150	3000	60
Filter	Mohr Coulomb	120	0	35

Because a probability analysis has been specified in the analysis settings the *Probabilistic* button is visible on the *Materials Manager* dialog. The probability parameters are specified as follows:

1. Select *Model > Materials > Manager...* from the menu if the *Materials Manager dialog* is not already open,

2. Click the *Probabilistic...* button to open the *Probabilistic Parameters* dialog,

3. Click the *Add/Remove...* button to open the *Add/Remove Probabilistic Parameters* dialog,

4. Expand the Phase I Fill and Phase II Fill tree items and check the *c* and *Phi* parameters for each item,

5. Click the *OK* button to close the dialog and populate the *Probability Parameters* data grid,

6. Enter the values shown in the table below for each material parameter,

Material	Property	Distribution	Mean	St. Dev.	Rel. Min	Rel. Max
Phase I Fill	c	Normal	2230	1150	2230	2230
Phase I Fill	Phi	Normal	6.33	7.87	6.33	6.33
Phase II Fill	c	Normal	2901.6	1107.9	2901.6	2901.6
Phase II Fill	Phi	Normal	14.8	9.44	14.8	14.8

7. Press the *OK* button to accept the changes and close the dialogs.

Now that all material properties have been entered, we must apply the materials to the corresponding regions.

1. Open the *Region Properties* dialog by selecting *Model > Geometry > Region Properties* from the menu,

2. For each region the appropriate material type must be selected from the combo box. The material assignments should be as follows:

   R1:   Phase II Fill

   R2:   Filter

   R3:   Phase I Fill

   R4:   Spoil Fill

   R5:   Material 3

   R6:   Material 3

   R7:   Material 4

3. Press the *OK* button to accept the changes and close the dialog.

g. Run Model (Solve > Analyze)
The next step is to analyze the model.

1. Select *Solve > Analyze* from the menu. A pop-up dialog will appear and the solver will start.

h. Visualize Results (Window > AcuMesh)
After the model has completed solving, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on the *Acumesh...* button. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen.

9.2   Results and Discussion

The critical slip surface for this numerical model is displayed when the model is first opened in ACUMESH by the user. The summarized probabilistic model results are presented in *Probability > Monte Carlo...* menu option. This dialog displays the normal distribution of the factor of safety. On the *Values* tab, the reliability index, the probability of failure, and a normal distribution of the probability of failure are given.
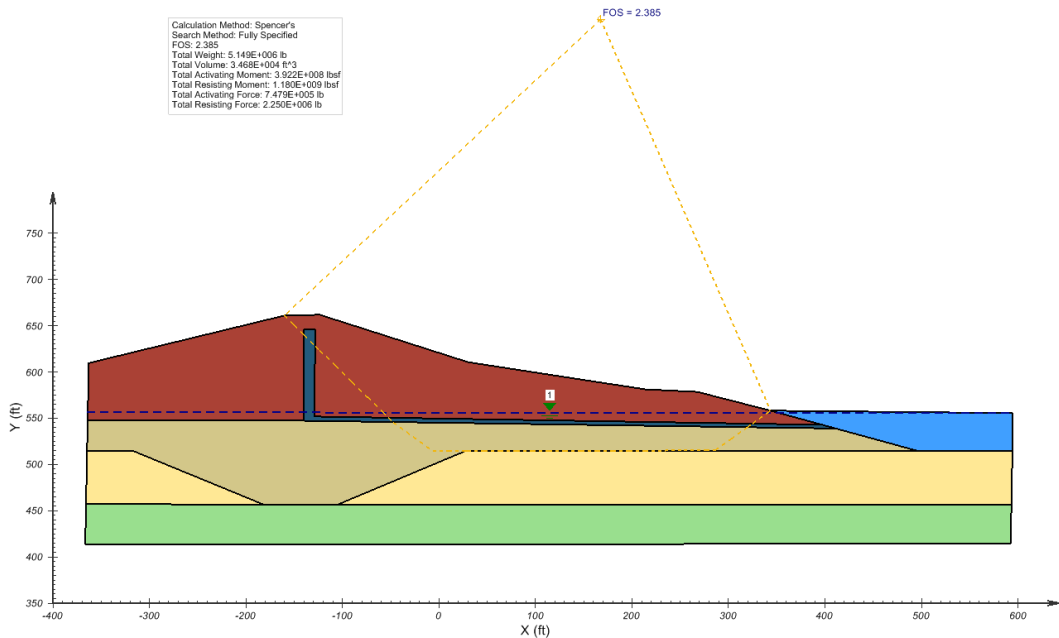
The probabilistic analysis results from SVSLOPE are compared to the results published in the paper by Wolff and Harr in the table below. The difference in factor of safety between the SVLOPE probabilistic model and the published values is less around 1% for both the Spencer and GLE calculation methods. The probability of failure is less than 1% for both SVSlope methods and the published value.

Method	Factor of Safety					Difference in FOS (%)
	Wolff and Harr		SVSLOPE			
				Probabilitistic		
	Deterministic	PF (%)	Deterministic	Mean	PF (%)	
Spencer	2.36	0.455	2.383	2.385	0.554	1.06
GLE			2.338	2.343	0.707	0.72

9.3   Model Data

Region Geometries

Region: R1

X (ft)	Y (ft)
-362.901	609.581
-364.066	547.255
-140.1	547.234
-140.1	646.256
-128.181	646.256
-129.097	551.818
-19.534	549.951
397.175	542.848
343.636	558.134
267.91	578.407
213.814	581.158
30.439	610.498
-123.596	661.843
-160.271	660.926

Region: R2

X (ft)	Y (ft)
-140.1	646.256
-140.1	547.234
411.261	538.826
397.175	542.848
-19.534	549.951
-129.097	551.818
-128.181	646.256

Region: R3

X (ft)	Y (ft)
-364.066	547.255
-364.683	514.226
-316.14	514.226
-181.359	456.463
-180.442	456.463
-105.259	456.463
26.771	514.226

497.42	514.226
411.261	538.826
-140.1	547.234

Region: R4

X (ft)	Y (ft)
343.636	558.134
397.175	542.848
411.261	538.826
497.42	514.226
594.268	514.226
594.317	516.977
594.317	555.486

Region: R5

X (ft)	Y (ft)
-364.683	514.226
-365.747	457.294
-181.359	456.463
-316.14	514.226

Region: R6

X (ft)	Y (ft)
-105.259	456.463
593.237	456.464
594.268	514.226
497.42	514.226
26.771	514.226

Region: R7

X (ft)	Y (ft)
-365.747	457.294
-366.568	413.37
592.484	414.287
593.237	456.464
-105.259	456.463
-180.442	456.463
-181.359	456.463

Return to Enter Geometry step

10   2D Spatial Variability Example

This tutorial illustrates a re-analysis of a classic model analyzed using spatial variability. The "Basic Slope" model is now re-analyzed and the differences to the classic solution are noted as parameters for the spatial variation of soil properties are assumed.

This original model can be found under:
Project: Slopes_Group_2
Model: VW_9

Minimum authorization required to complete this tutorial: FULL
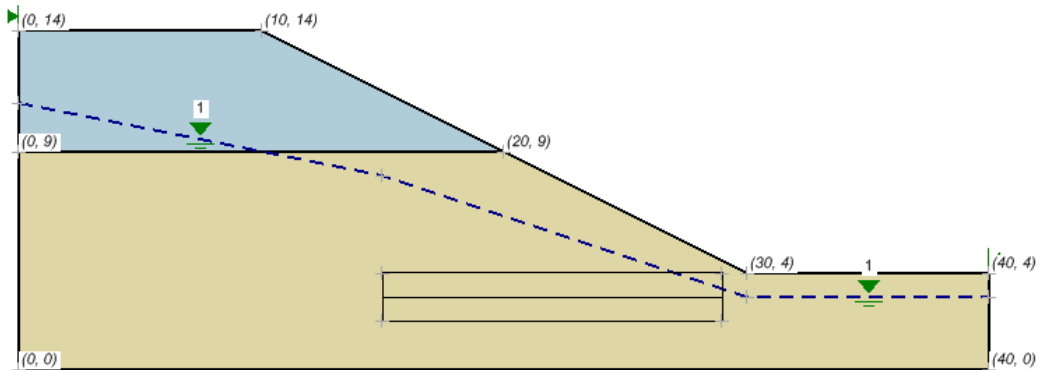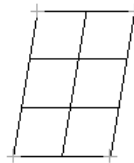
10.1   Model Setup

In order to set up this tutorial model, we will utilize the Basic Slope tutorial model and enable spatial variability in the analysis. The steps to create this model fall under the general categories of:

    a.   Create model

    b.   Specify spatial variability of material properties

    c.   Run model

    d.   Visualize results

The details of these outlined steps are detailed in the following sections.

> **N O T E :**
> Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

In order to create the spatial variability model, save a copy of the Basic Slope model. This is accomplished through the following steps:

1. Open the *SVOFFICE Manager* dialog,

2. Press the *Clear Search* button (if enabled),

3. Select the "UserTutorial" project and open the "Basic Slope" model, The may begin with the "VW_9" model under the "Slopes_Group_2 project" if the Basic Slope Example was not created,

4. Select *File > Save As...* from the menu,

5. Type the name "Spatial Variability Example" and click OK.

A new model has been created and loaded into the workspace that will be modified to include spatial variability analysis.

b. Specify Spatial Variability of Material Properties (Model > Materials > Spatial Variability)

The next step is to enable spatial variability. This is accomplished through the following steps:

1. Select *Model > Settings...* from the menu,

2. Select the spatial variability tab,

3. Select the *2D Spatial Variability* option with the following settings,

    Generator Seed: 500

    Covariance Function: dlavx2

4. Press *OK* to close the dialog.

Spatial variability allows a generation of random or user-specified fields of soil parameters (such as cohesion or friction angle) which vary spatially across any particular region. The next step is to generate a random field for all soil parameters for both regions in the model.

1. Open the *Spatial Variability Parameters* dialog by selecting *Model > Materials > Spatial Variability...* from the menu,

2. Select the random *Field Parameters* tab and press the *Add/Remove...* button,

3. Click the *Add All* button to add all parameters for all regions to the list of random field parameters,

4. Click *OK* to close the dialog,

5. The default values for the random field parameters are displayed in the data grid on the *Random Field Parameters* tab. To increase the spatial resolution of the random field parameters, enter the following values,

Number of Grid-X values for the Upper Soil: 50

Number of Grid-Y values for the Lower Soil: 100

6.   Click *OK* to close the *Spatial Variability Parameters* dialog,

A screenshot of the spatial variability contours for the lower region (R2) is shown below. The user may change the spatial variability contour settings by selecting *Model > Materials > Spatial Variability Contouring...* from the menu.

N O T E :
The spatial variability contours for both regions are not displayed at the same time because the random field parameters may differ significantly between the materials, in which case a single set of contours is not appropriate.
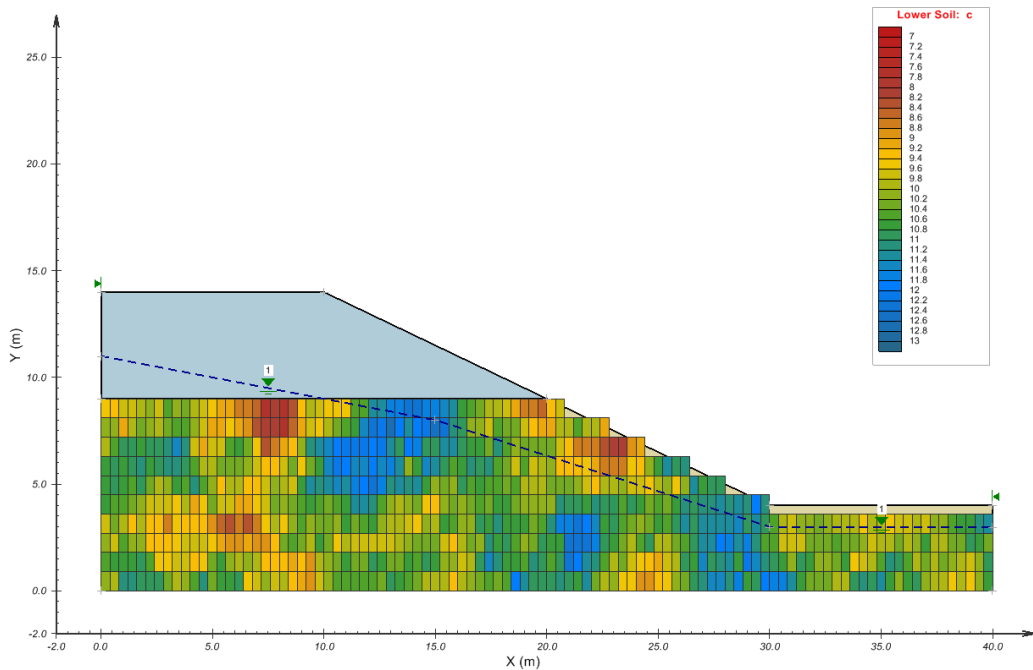
c.   Run Model (Solve > Analyze)

The next step is to analyze the model.

1.   Select *Solve > Analyze* from the menu. A pop-up dialog will appear and the solver will start.

d.   Visualize Results (Window > AcuMesh)

After the model has completed solving, an ACUMESH notification will appear asking if you

want to view the results in ACUMESH. Click on the *Acumesh...* button. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen.

10.2   Results and Discussion

The critical slip surface for the numerical model is displayed when the model is first opened in ACUMESH. The critical slip surface for Bishop's method is shown below. The FOS values have been increased for all calculation methods by using spatial variability in the model.
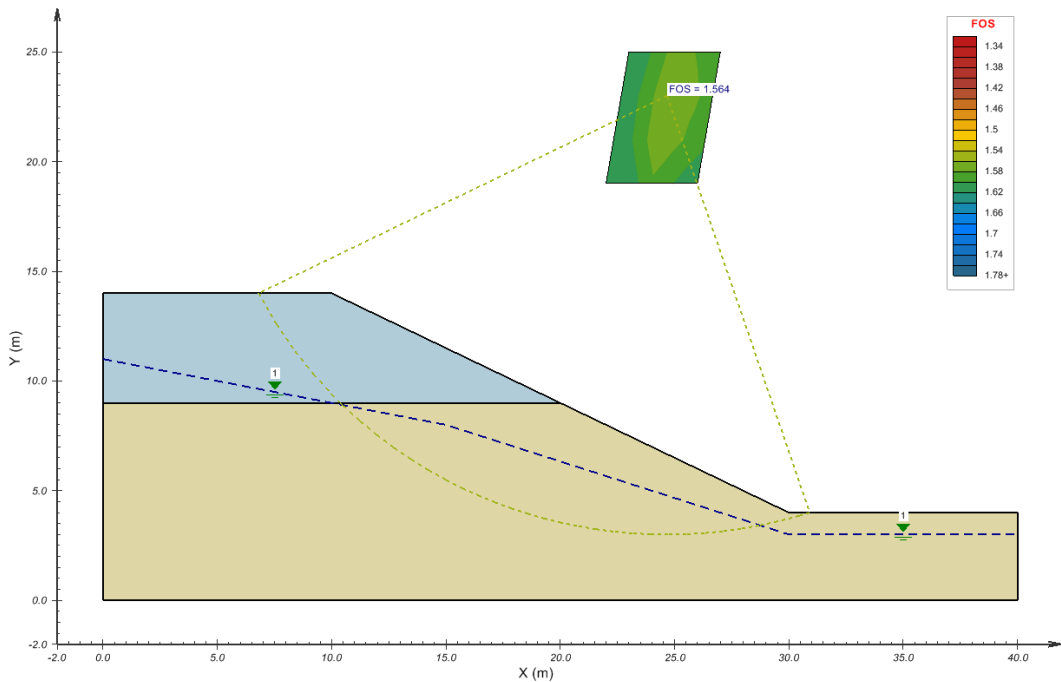
11   2D Two-Way Sensitivity Analysis

This tutorial illustrates a re-analysis of a classic model analyzed using a sensitivity analysis. The "Basic Slope" model is used as a starting point for the analysis and the relationship between cohesion and friction angle is analyzed.

This model can be found under:
Project: Slopes_Group_3
Model: VW_9__Sensitivity_1

Minimum authorization required to complete this tutorial: FULL
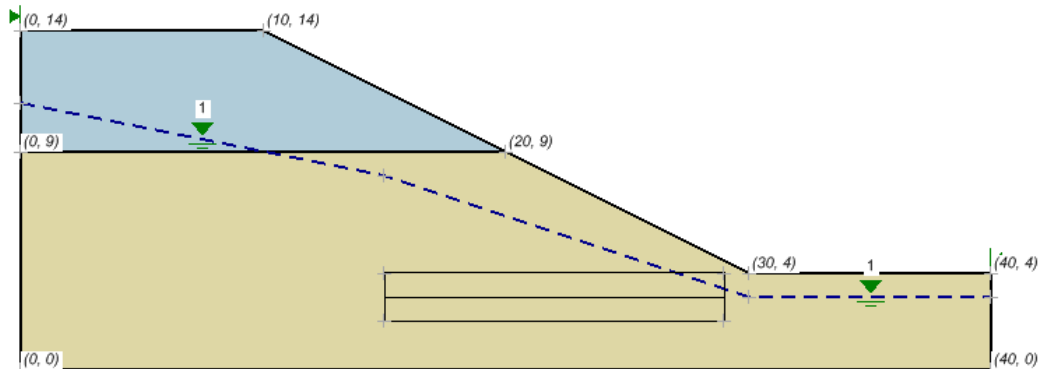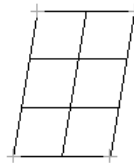
11.1   Model Setup

In order to set up this tutorial model, we will utilize the Basic Slope tutorial model and enable spatial variability in the analysis. The steps to create this model fall under the general categories of:

     a.   Create model

     b.   Specify sensitivity analysis

     c.   Specify sensitivity parameters

     d.   Run model

     e.   Visualize results

The details of these outlined steps are detailed in the following sections.

N O T E :

> Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

In order to create the sensitivity model, save a copy of the Basic Slope model. This is accomplished through the following steps:

1. Open the *SVOFFICE Manager* dialog,
2. Press the *Clear Search* button (if enabled),
3. Select the "UserTutorial" project and open the "Basic Slope" model. The may begin with the "VW_9" model under the "Slopes_Group_2 project" if the Basic Slope Example was not created,
4. Select *File > Save As...* from the menu,
5. Type the name "Sensitivity Example" and click OK.

A new model has been created and loaded into the workspace that will be modified to include a two-way sensitivity analysis.

b. Specify Sensitivity Analysis (Model > Settings)

The next step is to enable sensitivity analysis. This is accomplished through the following steps:

1. Select *Model > Settings...* from the menu,
2. Select the *Sensitivity/Probability* tab,
3. Select the Sensitivity Analysis option with the following settings,

     Sensitivity Parameters: Two-Way Sensitivity

     Critical Slip Surface Location: Floating
4. Press *OK* to close the dialog.

c. Specify Sensitivity Parameters (Model > Materials)

In a sensitivity analysis model input parameters are specified as varying to a range of properties for a given region in the model. Therefore, the model is run multiple times in increments as one or more of the values are changed in a logical progression. A two-way analysis was specified in the previous step. In the two way analysis, the user is allowed to vary two separate parameters in a logical fashion, such that the impact of the two parameters on the factor of safety can be determined. The *c* and *phi* parameters will be varied for region *R1* in this model by the following steps:

1. Select *Models > Materials > Manager...* from the menu,
2. Click the *Sensitivity...* button,
3. On the *Parameters* tab click the *Add/Remove...* button,

4. Expand the *R1* region and check the *c* and *phi* check boxes to include these material parameters in the sensitivity analysis. The default parameter ranges will be used and are shown in the following table,

Global/Material	Material	Property	Mean	Min	Max
Material	R1	c (kPa)	5	0	7.5
Material	R1	Phi (deg)	15	0	22.5

5. Move to the *Two-Way Pairs* tab,
6. Click on the data grid cell in the Parameter #1 column and click on the *Edit...* button,
7. Check the *c* parameter check box and click *OK* to close the dialog,
8. Click on the data grid cell in the Parameter #2 column and click on the Edit... button
9. Check the Phi parameter check box and click *OK* to close the dialog,
10. The list of parameter values for each model run can be seen on the *Run List* tab and should appear as follows,

Run #	R1 c (kPa)	R1 Phi (deg)
1	0	15
2	1.667	15
3	3.333	15
4	5	15
5	5.833	15
6	6.667	15
7	7.5	15
8	5	0
9	5	5
10	5	10
11	5	15
12	5	17.5
13	5	20
14	5	22.5

11. Click *OK* to close the *Materials Manager* dialog.

c. Run Model (Solve > Analyze)

The next step is to analyze the model.

1. Select *Solve > Analyze* from the menu. A pop-up dialog will appear and the solver will start.

d. Visualize Results (Window > AcuMesh)

After the model has completed solving, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on the *Acumesh...* button. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen.

11.2 Results and Discussion

The results of performing a sensitivity analysis can be seen under the menu item *Sensitivity*. The *Plots* dialog shows how the factor of changes by varying each parameter individually. The *Two-Way Sensitivity Contour* dialog shows the factor of safety for each combination of sensitivity parameter values. The screenshot below shows that the factor of safety decreases with decreasing *c* and *phi* values and increases with increasing *c* and *phi* values in this model.
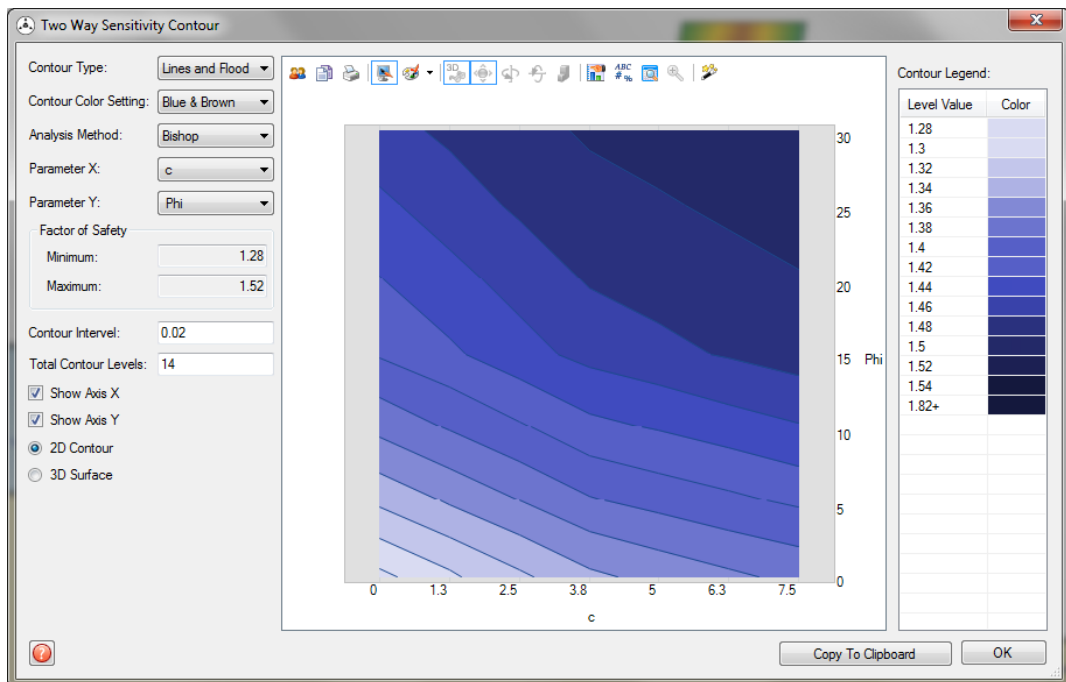
12   3D Multi Planar Example

The following example will introduce you to the three-dimensional SVSLOPE modeling environment. This example is used to investigate the use of a wedge slip surface method in determining the critical slip surface. A simple geometry is utilized in this example which is extruded from a 2D cross-section.

This original model can be found under:

Project: Slopes_3D
Model: Multi_Planar_Wedges

Minimum authorization required: STANDARD

Model Description and Geometry

A simple 120m by 180m area is created. A non-level plane is added to model the ground surface. A triangular pile is then added to the flat ground surface.
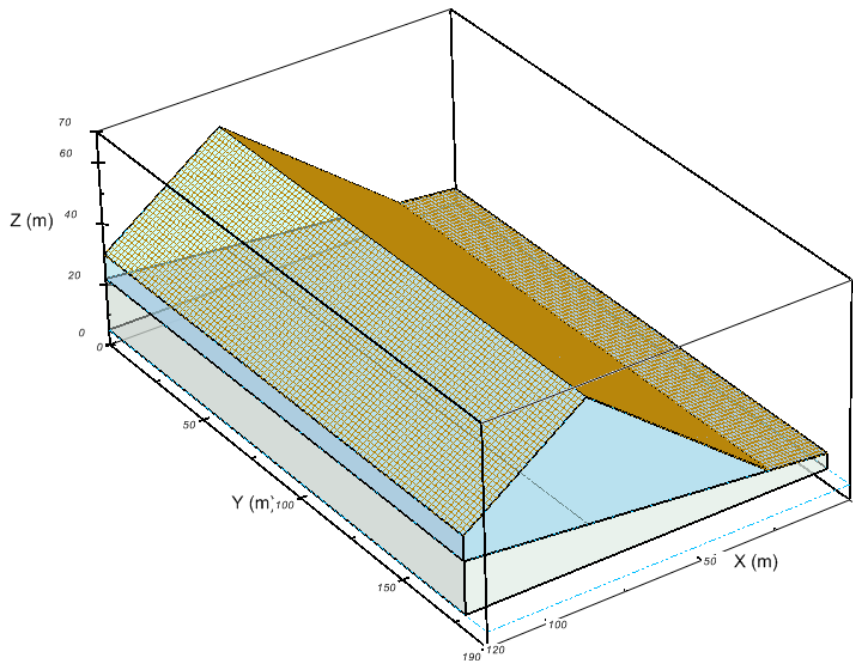
12.1   Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the following general categories:

       a.   Create model

       b.   Enter geometry

       c.   Specify pore water pressure

       d.   Apply material properties

   e.   Extrude 2D model to 3D

   f.   Specify analysis settings

   g.   Specify search method geometry

   h.   Run model

   i.   Visualize results

The details of these outlined steps are given in the following sections.

> **N O T E :**
> Any values on the dialogs that are not specifically mentioned in the steps below are
> assumed to be the default values currently present.

a. Create Model

STANDARD authorization is required for this tutorial. The steps to ensure that STANDARD
authorization is activated are described in the Authorization section.

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,

2. Press the Clear Search button (if enabled),

3. Create a new project called "UserTutorial" by pressing the *New...* button next to
   the list of projects,

4. Create a new model called "UserMultiWedge2D" by pressing the *New...* button
   next to the list of models. Note that initially the model is constructed as a 2D
   model to be later extruded to a 3D model. Use the settings below when creating
   this new model:

Application:	SVSLOPE
Model Name:	UserMultiWedge2D
System:	2D
Units:	Metric
Slip Direction:	Right to Left

5. Click on the *World Coordinate System* tab,

6. Enter the World Coordinates System coordinates shown below into the dialog
   (leave Global Offsets as zero),

x min = 0	x max = 120
y min = 0	y max = 70

7. Click on *OK*.

The new model will be automatically added under the recently created UserTutorial project.

SVSLOPE now opens to show a grid and the Options dialog (*View > Options)* pops up. Click
OK to accept the default horizontal and vertical grid spacing of 1.0.

b. Enter Geometry (Model > Geometry)

Model geometry is defined as a set of regions and a series of layers. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from either .DXF files or from existing models.

The shapes that define each region will now be created. Refer to the two data tables below for the geometry points for each region.

- **Define Region R1**

    1.   Select *R1* from the Region Selector,
    2.   Select *Draw > Geometry > Polygon Region* from the menu,
    3.   The cursor will now be changed to a cross hair,
    4.   Move the cursor near (120,5) in the drawing space,
    5.   To select the point as part of the shape left click on the point,
    6.   Now move the cursor near (0,5) and left-click the mouse. A line is now drawn from (120,5) to (0,5),
    7.   In the same manner then enter the following points:
         (0,10)
         (20,12)
         (80,18)
    8.   Move the cursor near the point (120,22). Double click on the point to finish the shape. The shape is automatically finished by SVSLOPE by drawing a line from (120,22) back to the start point (120,5).

Repeat this process to define the region *R2* using the data provided in the table below.

Region 1: R1

X (m)	Y (m)
120	5
0	5
0	10
20	12
80	18
120	22

Region 2: R2

X (m)	Y (m)
120	22
80	18
20	12
80	58
120	30

NOTE:
If an error is made when entering the region geometry the user may recover from the error and start again by one of the following methods:

a. Press the escape (esc) key.
b. Select a region shape and press the delete key.
c. Use the Undo function on the Edit menu.

c. Specify Pore Water Pressure (Model > Pore Water Pressure)

Generally information will be entered either for a water table or a piezometric line. In this model a water table will be used. In order to specify that a water table will be entered the must perform the following steps:

1. Select *Model > Pore Water Pressure > Settings...*,

2. Select "Water Surfaces" as the Pore-Water Pressure Method,

3. Check the "Allow application of RU coefficients with water surfaces or discrete points" checkbox,

4. Press *OK* to close the dialog.

The user must then proceed to enter the water table coordinates:

1. Select *Model > Pore Water Pressure > Water Table...*,

2. On the *Points* tab enter the *X* and *Y* coordinates as provided in the table below by copying the values (excluding the header) and clicking the *Paste Points* button on the dialog,

3. Press *OK* to close the dialog.

X (m)	Y (m)
0	10
20	12
80	18
120	22

d. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the three materials that will be used in the model. The bottom region represents the foundation and will be assigned a clay material. The top region represents a pile placed above the foundation and will be assigned a fill material. An additional material is created to later define a wedge corresponding to a discontinuity. This section will provide instructions on creating the *Fill* material. Repeat the process to add the other two materials.

1. Open the *Materials Manager* dialog by selecting *Model > Materials > Manager...* from the menu,

2. Click the *New...* button to create a material,

3. Enter "Fill" for the material name in the dialog that appears and choose Mohr Coulomb for the Shear Strength type of this material,

4. Press OK to close the dialog. The Material Properties dialog will open automatically,

5. Move to the Shear Strength tab and enter the parameter values provided in the table below,

6. Move to the Water Parameters tab,

7. Select "Off" for the Water Surfaces,

8. Enter a *Ru Coefficient* value of 0.4,

9. Click the OK button to close the Mohr Coulomb dialog,

10. Repeat these steps to create the *Clay Foundation* and *Disc* materials using the information provided at the beginning of the tutorial,

11. Press the *OK* button on the *Materials Manager* dialog to accept the changes and close the dialog.

Material	Shear Strength Type	Cohesion (kPa)	Friction Angle (deg)	Unit Weight (kN/m³)	Water Surfaces	Ru Coefficient
Fill	Mohr Coulomb	0	35	18	Off	0.4
ClayFoundation	Mohr Coulomb	50	20	20	On	-
Disc	Mohr Coulomb	0	12	0.001	On	-

Once all three material properties have been entered, we must apply the materials to the corresponding regions.

1. Open the *Region Properties* dialog by selecting *Model > Geometry > Region Properties...* from the menu,

2. Select the *R1* region using the arrows at the top right of the dialog.

3. Under the "Region Settings" group select the *Clay Foundation* material from the combo box to assign this material to *R1*,

4. Select the *R2* region and assign the *Fill* material to this region,

5. Press the *OK* button to accept the changes and close the dialog.

e. Extrude 2D Model to 3D (File > Save As)

All of the previous steps may be transferred to a 3D version of this model. A new model is created with 3D geometry by extruding the 2D cross-section from the current model. This is accomplished through the following steps:

> **N O T E :**
> At this point the user may wish to analyze the 2D model to examine the factor of safety. This process is described in the steps h and i below.

1. First, save the current model by clicking *File > Save* from the menu,

2. Next, to begin the extrusion process select *File > Save As...* from the menu,

3. Select the *General* tab,

System: 3D

New File Name: User Multi Planar 3D

4.   Select the *Spatial* tab,
5.   Enter the following model extrusion parameters,

Y minimum: 0 m
Y maximum: 180 m

6.   Press *OK* to close the dialog,
7.   Press *OK* to accept the reset of some items,
8.   Select *View > Mode > 3D* to change the CAD to a 3D view.

> **N O T E :**
> X- and Y-coordinates in 2D become X- and Z-coordinates in 3D space with the model extrusion

The 3D geometry is now complete.

f. Specify Analysis Settings (Model > Settings)

The Analysis Settings provide the information for what type of analysis will be performed. These settings will be specified by the following steps:

1.   Select *Model > Settings...* from the menu,
2.   Select the *3D Slip Surface* tab,
     Search Method: Fully Specified - Wedges
3.   Select the *Calculation Methods* tab from the dialog and select the method types as shown below:
     Bishop Simplified
     Spencer
     Morgenstern-Price
     GLE (Fredlund)
4.   For GLE method, press the *Lambda...* button,
5.   Enter a Start Value of -0.5, an Interval of 0.25, and a Number of 8,
6.   Press the *Generate* button,
7.   Press *OK* to close the dialogs.

g. Specify Search Method Geometry (Model > Slip Surface)

This model makes use of a fully-specified search methodology. The wedge shape is used as the slip surface geometry. Four wedges will be specified through the following steps:

1.   Open the *Wedges Sliding Surface* dialog through the *Model > Slip Surface > Fully Specified > Wedges...* menu option,

2. Enter the data for the 4 wedges as specified below by copying the data (do not include the header row) and clicking the *Paste Points* button on the dialog,

3. Click *OK* to close the dialog.

X (m)	Y (m)	Z (m)	Dip (deg)	Dip Direction	Discontinuity Material
0	90	10	7	0	Disc
60	90	12	32	0	None
0	90	-35	45	87	None
0	90	-35	45	-87	None

h. Run Model (Solve > Analyze)

The next step is to analyze the model.

4. Select Solve > Analyze from the menu. A pop-up dialog will appear and the solver will start.

i. Visualize Results (Window > AcuMesh)

After the model has completed solving, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on the *Acumesh...* button. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen.

12.2 Results and Discussions

After the model has completed solving the user may view the results in the ACUMESH software by pressing the ACUMESH icon on the process toolbar. The sliding mass is displayed in the CAD for the selected calculation method. To switch between the results of the different calculation methods, click on the drop down menu at the top of the screen and select the method you would like to view. In order to view the sliding mass area more clearly the user may edit the *Critical Sliding Mass* dialog:

1. Select *Slips > Critical Sliding Mass...* from the menu,

2. Under the *Mass Explosion* tab uncheck the *Show Sliding Mass with Explosion under 3D View* checkbox.

The analysis results in a factor of safety of 1.172 for the Spencer method.

Calculation Method: Spencer's
Search Method: Fully Specified - Wedges
FOS: 1.172
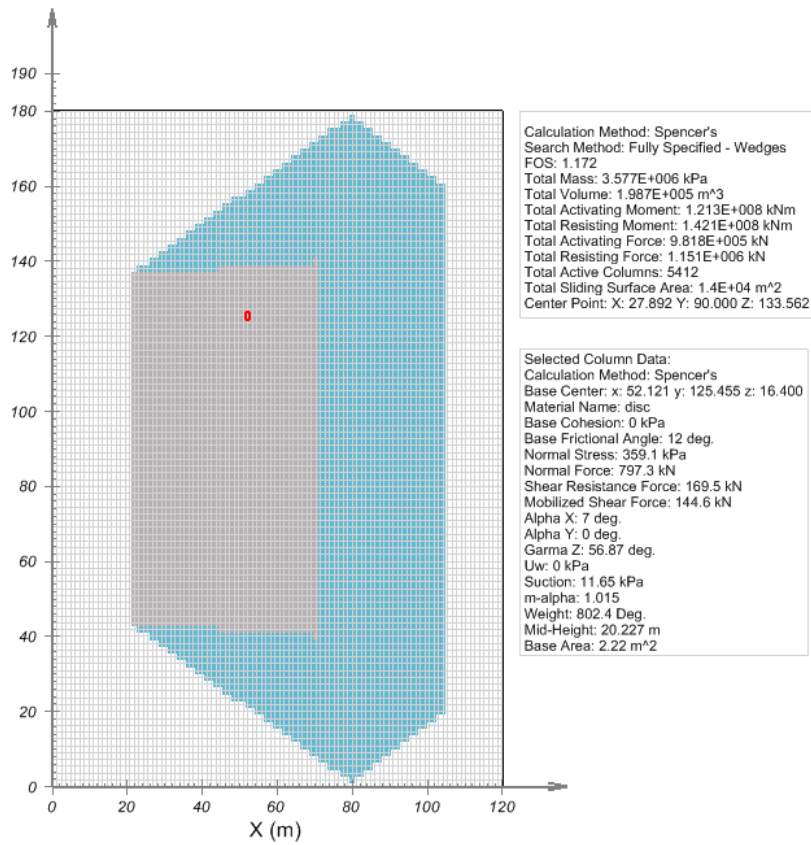Total Mass: 3.577E+006 kPa
Total Volume: 1.987E+005 m^3
Total Activating Moment: 1.213E+008 kNm
Total Resisting Moment: 1.421E+008 kNm
Total Activating Force: 9.818E+005 kN
Total Resisting Force: 1.151E+006 kN
Total Active Columns: 5412
Total Sliding Surface Area: 1.4E+04 m^2
Center Point: X: 27.892 Y: 90.000 Z: 133.562
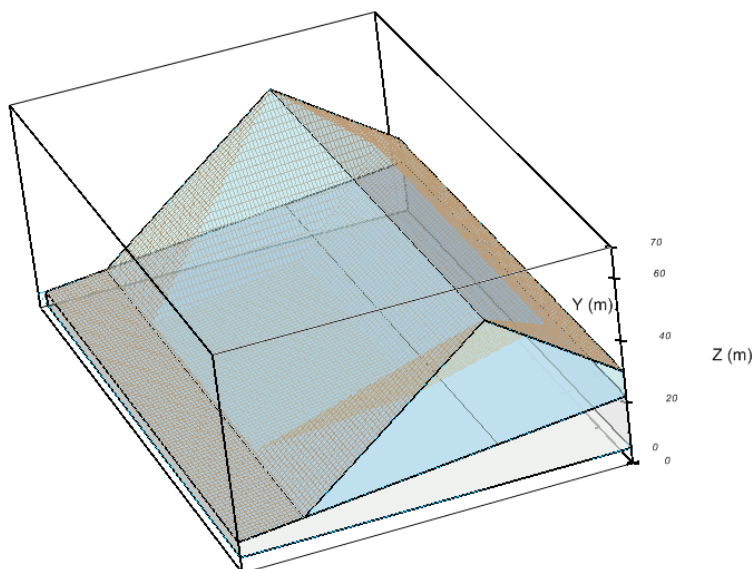
The user may also plot the column information for a particular column chosen either in plan view or from a vertical cross-section. The column information settings are set in the *Column Information* dialog. To access this dialog, first click on a 2D view of the model by selecting one of the 2D options under *View > Mode*. Then click the *Slips > Column Information...* menu item. Once the *Column Information* dialog is closed by clicking *OK*, the user may select a particular column by clicking on it in the CAD. The details of the selected column will appear in the CAD.

Calculation Method: Spencer's
Search Method: Fully Specified - Wedges
FOS: 1.172
Total Mass: 3.577E+006 kPa
Total Volume: 1.987E+005 m^3
Total Activating Moment: 1.213E+008 kNm
Total Resisting Moment: 1.421E+008 kNm
Total Activating Force: 9.818E+005 kN
Total Resisting Force: 1.151E+006 kN
Total Active Columns: 5412
Total Sliding Surface Area: 1.4E+04 m^2
Center Point: X: 27.892 Y: 90.000 Z: 133.562

Selected Column Data:
Calculation Method: Spencer's
Base Center: x: 52.121 y: 125.455 z: 16.400
Material Name: disc
Base Cohesion: 0 kPa
Base Frictional Angle: 12 deg.
Normal Stress: 359.1 kPa
Normal Force: 797.3 kN
Shear Resistance Force: 169.5 kN
Mobilized Shear Force: 144.6 kN
Alpha X: 7 deg.
Alpha Y: 0 deg.
Garma Z: 56.87 deg.
Uw: 0 kPa
Suction: 11.65 kPa
m-alpha: 1.015
Weight: 802.4 Deg.
Mid-Height: 20.227 m
Base Area: 2.22 m^2

13   3D Submergence Example

The following example is used to illustrate the use of a grid and tangent search method in determining the critical slip surface of a submerged slope. The example is modeled using four regions, five surfaces, and four materials. A simple geometry is utilized in this example which is extruded from a 2D cross-section.

This original model can be found under:

Project: Slopes_3D
Model: Grid_Tangent_Toe_Submergence

Minimum authorization required: STANDARD

Model Description and Geometry

A simple 500ft by 200ft area is created. A non-level plane is added to model the underlying bedrock layer. The slope shape lying above bedrock is composed of 3 surfaces. The top surface intersects the water table at a height 150.07ft.
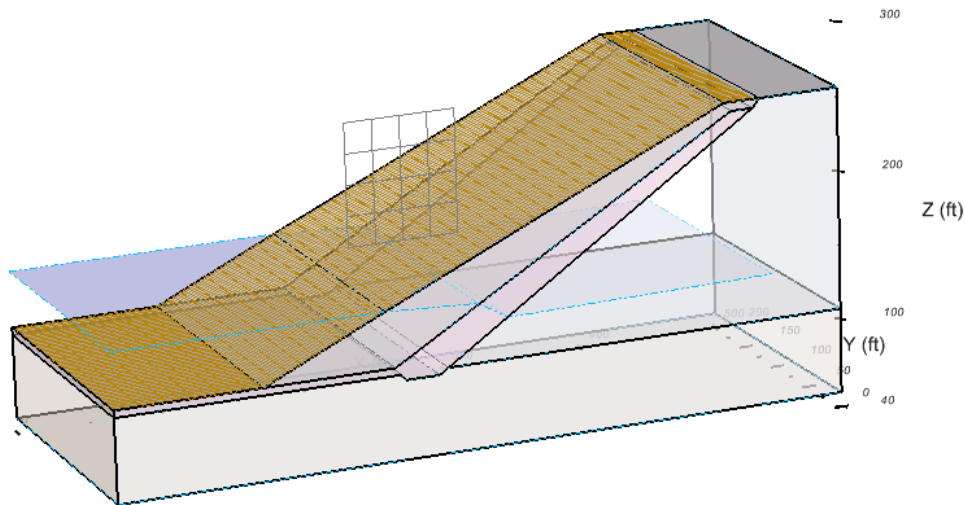
13.1   Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the following general categories:

  a. Create model

  b. Specify analysis settings

  c. Enter geometry

  d. Specify pore water pressure

  e. Apply material properties

  f. Specify search method geometry

    g.   Extrude 2D model to 3D

    h.   Run model

    i.   Visualize results

The details of these outlined steps are given in the following sections.

> **N O T E :**
> Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

STANDARD authorization is required for this tutorial. The steps to ensure that STANDARD authorization is activated are described in the Authorization section.

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,
2. Press the Clear Search button (if enabled),
3. Create a new project called "UserTutorial" by pressing the *New...* button next to the list of projects,
4. Create a new model called "User Submergence 2D" by pressing the *New...* button next to the list of models. Note that initially the model is constructed as a 2D model to be later extruded to a 3D model. Use the settings below when creating this new model:

       Application: SVSLOPE

       Model Name: User Submergence 2D

       System: 2D

       Units: Imperial

       Slip Direction: Right to Left

5. Click on the *World Coordinate System* tab,
6. Enter the World Coordinates System coordinates shown below into the dialog,

       x min = 0   x max = 300

       y min = 0   y max = 550

7. Click on *OK*.

The new model will be automatically added under the recently created UserTutorial project.

SVSLOPE now opens to show a grid and the Options dialog (*View > Options)* pops up. Change the default horizontal and vertical grid spacing to 1.0 ft and click *OK*.

b. Specify Analysis Settings (Model > Settings)

In SVSlope the Analysis Settings provide the information for what type of analysis will be performed. These settings will be specified as follows:

1. Select *Model > Settings* from the menu,
2. Select the Slip Surface tab,

      Slip Direction: Right to Left

      Slip Shape: Circular

      Search Method: Grid and Tangent

3. Select the *Calculation Methods* tab from the dialog and select the method types as shown below:

      Bishop Simplified

      Janbu Simplified

4. Select the *Convergence* tab from the menu,

      Number of slices: 50

5. Press OK to close the dialog.

c. Enter Geometry (Model > Geometry)

Model geometry is defined as a set of regions and a series of layers. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from either .DXF files or from existing models.

The shapes that define each region will now be created. Refer to the tables below for the geometry points for the four regions.

- **Define Region R1**

1. Select *R1* from the Region Selector,
2. Select *Draw > Geometry > Polygon Region* from the menu,
3. The cursor will now be changed to a cross hair,
4. Move the cursor near (500,50) in the drawing space,
5. To select the point as part of the shape left click on the point,
6. Now move the cursor near (0,50) and left-click the mouse. A line is now drawn from (500,50) to (0,50),
7. In the same manner then enter the following points:

    (0,107)

    (193,107)

    (196,103)

    (219,103)

    (225,107)

8. Move the cursor near the point (500,107). Double click on the point to finish the shape. A line is now drawn from (225,107) to (500,107). The shape is automatically finished by SVSLOPE by drawing a line from (500,107) back to the start point (500,50).

Repeat this process to define the regions *R2*, *R3* and *R4*.

N O T E :

If an error is made when entering the region geometry the user may recover from the error and start again by one of the following methods:
    a. Press the escape (esc) key.
    b. Select a region shape and press the delete key.
    c. Use the Undo function on the Edit menu.

Region 1: R1

X (ft)	Y (ft)
500	50
0	50
0	107
193	107
196	103
219	103
225	107
500	107

Region 2: R2

X (ft)	Y (ft)
500	107
225	107
440	250
445	256
500	256

Region 3: R3

X (ft)	Y (ft)
440	250
225	107
219	103
196	103
193	107
0	107
0	112
100	112
189	112
425	250

Region 4: R4

X (ft)	Y (ft)
445	256
440	250
425	250
189	112
100	112
420	256

d. Specify Pore Water Pressure (Model > Pore Water Pressure)

Initial conditions are generally associated with transient model runs. Their purpose is to provide a reasonable starting point for the solver. In a steady-state model, initial conditions can be used to "precondition" the solver to allow faster convergence. Generally speaking, the user will enter information either for a water table or a piezometric line. In this model a water table will be used. In order to specify that a water table will be entered the user must perform the following steps:

1. Select Model > *Pore Water Pressure* > Settings…,
2. Select "Water Surfaces" as the Pore-Water Pressure Method,
3. Press *OK* to close the dialog.

The user must then proceed to enter the water table coordinates:

1. Select *Model > Pore Water Pressure > Water Table…* from the menu,
2. Under the *Points* tab enter the *X* and *Y* coordinates as provided in the table below by copying the values (excluding the header) and clicking the *Paste Points* button on the dialog,
3. Press *OK* to close the dialog.

X (ft)	Y (ft)
0	150.012
184.472	150.012
247.595	146.263
266.837	134.826
451.8	137.39
500	137.39

> **N O T E :**
> After water table points are entered or a water table is drawn, the points will be automatically adjusted based on intersections of the water table line with regions. Corresponding points will be added to both the water table line and the regions.

e. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the four materials that will be used in the model. The *Extents* region cuts through all the surfaces in a model, creating a separate "block" on each layer. Each block can be assigned a material or be left as void. A void area is essentially air space. In this model all blocks will be assigned a material. There are five surfaces resulting in four layers. Each layer will contain a different material. This section will provide instructions on creating the *R1* material. Repeat the process to add the other three materials.

1. Open the *Materials* dialog by selecting *Model > Materials > Manager…* from the menu,
2. Click the *New…* button to create a material,
3. Enter "R1" for the material name in the dialog that appears and choose Mohr

Coulomb for the Shear Strength type of this material,

4. Press *OK* to close the dialog. The Material Properties dialog will open automatically,

5. On the Shear Strength tab and enter the parameter values provided in the table below,

6. Click the *OK* button to close the Mohr Coulomb dialog,

7. Repeat these steps to create the *Fill*, *Core*, and *RockFill* materials,

8. Press the *OK* button on the *Materials Manager* dialog to accept the changes and close the dialog.

Material	Shear Strength Type	Cohesion (psf)	Friction Angle (deg)	Unit Weight (lb/ft³)
R1	Mohr Coulomb	10000	35	100
RockFill	Mohr Coulomb	100	33	70.6
Core	Mohr Coulomb	10	29	70.6
Fill	Mohr Coulomb	0	28	70.6

Once all material properties have been entered, we must apply the materials to the corresponding regions.

1. Open the *Region Properties* dialog by selecting *Model > Geometry > Region Properties...* from the menu,

2. Select the *R1* region using the arrows at the top right of the dialog.

3. Under the "Region Settings" group select the *R1* material from the combo box to assign this material to region *R1*,

4. Select the *R2* region and assign the *Fill* material to this region,

5. Select the *R3* region and assign the *Core* material to this region,

6. Select the *R4* region and assign the *RockFill* material to this region,

7. Press the *OK* button to accept the changes and close the dialog.

f. Specify Search Method Geometry (Model > Slip Surface)

The Grid and Tangent method of searching for the critical slip surface has already been selected in Step b. Now the user must draw the graphical representation of the grid and tangent objects on the screen. This is accomplished through the following steps:

GRID

1. Select *Model > Slip Surface > Grid and Tangent...,*

2. Select the *Grid* tab,

3. Enter the values for the grid as provided in the table below (the grid values may also be drawn on the CAD window),

4. Move to entering the tangent values.

X (ft)	Y (ft)	
160	276	**Upper Left**
160	195	**Lower Left**
234	195	**Lower Right**

X Increments: 4
Y Increments: 4

TANGENT

1. Select the *Tangent* tab,
2. Enter the values for the tangent as provided in the table below (the grid values may also be drawn on the CAD window),
3. Press *OK* to close the dialog,
4. Accept the warning message stating that the corner of the grid is below the top of the slope,
5. Press *OK* to close the dialog.

X (ft)	Y (ft)	
0	128	**Upper Left**
0	108	**Lower Left**
500	108	**Lower Right**
500	128	**Upper Right**

Radius Increments: 1

The grid and tangent graphics should now be displayed on the CAD window.

g. Extrude 2D Model to 3D (File > Save As)

All of the previous steps may be transferred to a 3D version of this model. A new model is created with 3D geometry by extruding the 2D cross-section from the current model. This is accomplished through the following steps:

1. First, save the current model by clicking *File > Save* from the menu,

> **N O T E :**
> At this point the user may wish to analyze the 2D model to examine the factor of safety. This process is described in the steps h and i below.

2. Next, to begin the extrusion process select *File > Save As...* from the menu,
2. Select the *General* tab,

   System: 3D

   New File Name: User Submergence 3D

3. Select the *Spatial* tab,
4. Enter the following model extrusion parameters,

Y minimum: 0 ft

Y maximum: 200 ft

5. Press *OK* to close the dialog,

6. Press *OK* to accept the reset of some items.

7. Select *View > Mode > 3D* to change the CAD to a 3D view.

> **N O T E :**
> X- and Y-coordinates in 2D become X- and Z-coordinates in 3D space with the model extrusion

The Y-coordinates for the search method geometry need to be updated in the 3D model:

8. Select *Model > Slip Surface > Grid and Tangent...*,

9. Enter the following values for the Y-coordinate,

Min Value: 0

Max Value: 0

No. of Points: 1

The slope limits also need to be reset in the new 3D geometry:

10. Select *Model > Slope Limits*,

Min x: 0 ft   Max x: 441 ft

Min y: 0 ft   Max y: 200 ft

11. Press *OK* to close the dialog.

The 3D model is now complete and ready to be analyzed. The view may be switched between 2D and 3D mode by using the *View > Mode* menu item.

h. Run Model (Solve > Analyze)
The next step is to analyze the model.

1.   Select *Solve > Analyze* from the menu. A pop-up dialog will appear and the solver will start.

i. Visualize Results (Window > AcuMesh)
After the model has completed solving, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on the *Acumesh...* button. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen.
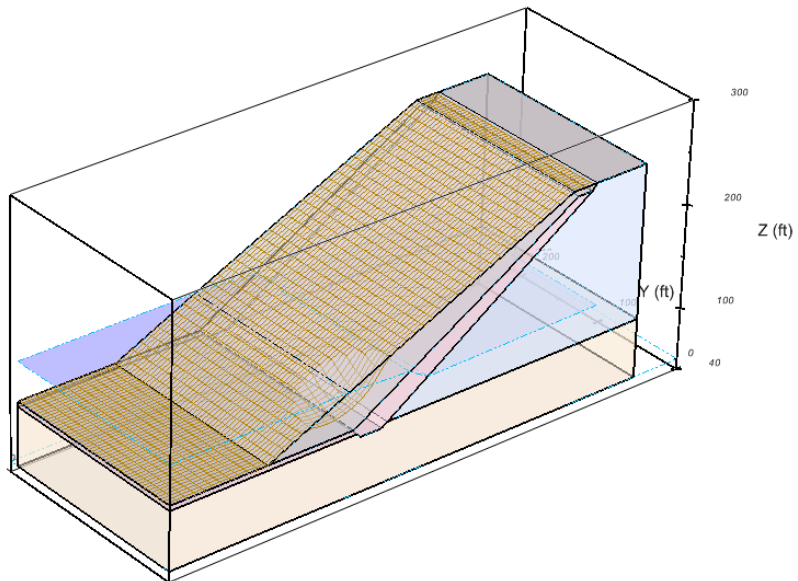
13.2   Results and Discussions
After the model has completed solving the user may view the results in the ACUMESH software by pressing the ACUMESH icon on the process toolbar. The sliding mass is displayed in the CAD for the selected calculation method. To switch between the results of the different calculation methods, click on the drop down menu at the top of the screen and

select the method you would like to view. In order to view the sliding mass area more clearly the user may edit the *Critical Sliding Mass* dialog:
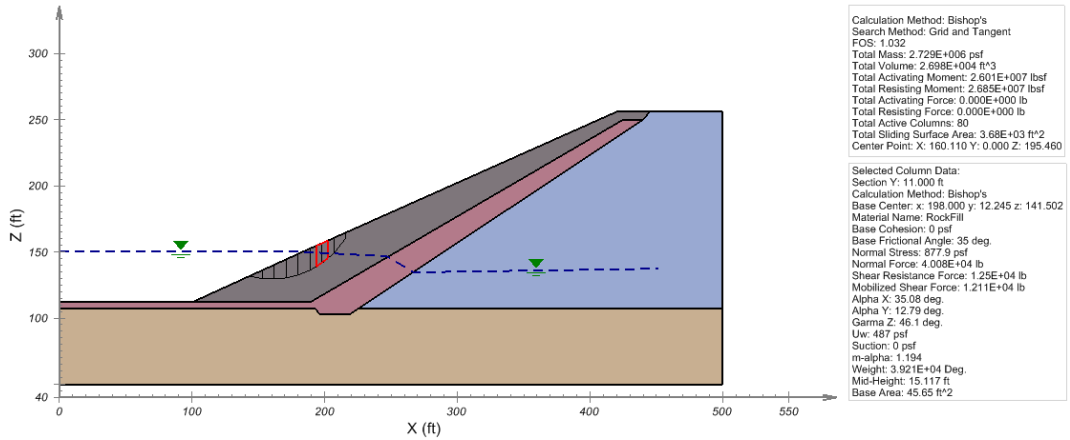
1. Select *Slips > Critical Sliding Mass...* from the menu,

2. Uncheck the *Sliding Mass with Explosion* checkbox.

The analysis results in a factor of safety of 1.032 for Bishop's method.

The user may also plot the column information for a particular column chosen either in plan view or from a vertical cross-section. The column information settings are set in the *Column Information* dialog. To access this dialog click the *Slips > Column Information...* menu item. Once the *Column Information* dialog is closed by clicking *OK*, the user may select a particular column by clicking on it in the CAD. The details of the selected column will appear in the CAD.

Calculation Method: Bishop's
Search Method: Grid and Tangent
FOS: 1.032
Total Mass: 2.729E+006 psf
Total Volume: 2.698E+004 ft^3
Total Activating Moment: 2.601E+007 lbsf
Total Resisting Moment: 2.685E+007 lbsf
Total Activating Force: 0.000E+000 lb
Total Resisting Force: 0.000E+000 lb
Total Active Columns: 80
Total Sliding Surface Area: 3.68E+03 ft^2
Center Point: X: 160.110 Y: 0.000 Z: 195.460

Selected Column Data:
Section Y: 11.000 ft
Calculation Method: Bishop's
Base Center: x: 198.000 y: 12.245 z: 141.502
Material Name: RockFill
Base Cohesion: 0 psf
Base Frictional Angle: 35 deg.
Normal Stress: 877.9 psf
Normal Force: 4.008E+04 lb
Shear Resistance Force: 1.25E+04 lb
Mobilized Shear Force: 1.211E+04 lb
Alpha X: 35.08 deg.
Alpha Y: 12.79 deg.
Garma Z: 46.1 deg.
Uw: 487 psf
Suction: 0 psf
m-alpha: 1.194
Weight: 3.921E+04 Deg.
Mid-Height: 15.117 ft
Base Area: 45.65 ft^2

14   3D General Sliding Surface

The following example is used to illustrate the use of a general sliding surface search method in determining the critical slip surface. The example is modeled using one region, three surfaces, and two materials. The purpose of this model is to demonstrate the entry of three-dimensional surfaces defined as elevation data.

This original model can be found under:

Project: Slopes_3D
Model: General_Sliding_Surface

Minimum authorization required: STANDARD

Model Description and Geometry

A simple 680 m by 500 m area is created. Two surfaces that pinch-out form a wedge-like layer that contains the glacial till material. The remainder of the model is composed of a waste rock material. A water surface exits near the midpoint of the slope.
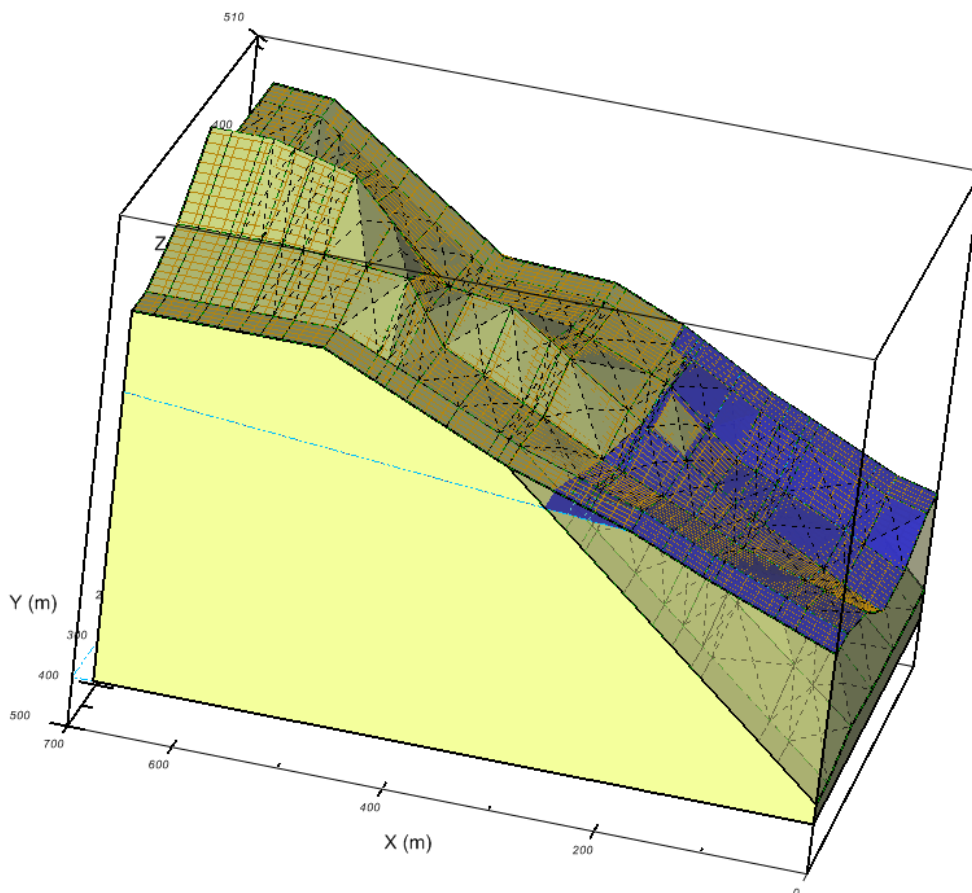
14.1   Model Setup

In order to set up the model described in the preceding section, the following steps are required. The steps fall under the following general categories:

    a.   Create model

    b.   Specify analysis settings

    c.   Enter geometry

    d.   Specify pore water pressure

    e.   Apply material properties

    f.   Specify search method geometry

    g.   Run model

    h.   Visualize results

The details of these outlined steps are given in the following sections.

> **N O T E :**
> Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

STANDARD authorization is required for this tutorial. The steps to ensure that STANDARD authorization is activated are described in the Authorization section.

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,
2. Press the Clear Search button (if enabled),
3. Select an existing project in the project list box or create a new project called "UserTutorial", for example, by pressing the *New...* button next to the list of projects,
4. Create a new model called "User General Sliding Surface 3D" by pressing the *New...* button next to the list of models. Use the settings below when creating this new model:

       Module: SVSLOPE

       Model Name: User General Sliding Surface 3D

       System: 3D

       Units: Metric

       Slip Direction: Right to Left

5. Click on the *World Coordinate System* tab and enter the World Coordinates System coordinates shown below,

       x min = 0   x max = 700

       y min = 0   y max = 500

       z min = 0   z max = 510

6.   Click on *OK*.

The new model will be automatically added under the selected project.

SVSLOPE now opens to show a grid and the Options dialog (*View > Options)* pops up. Change the default horizontal and vertical grid spacing to 1.0 m and click *OK*.

b. Specify Analysis Settings (Model > Settings)

In SVSlope the Analysis Settings provide the information for what type of analysis will be performed. These settings are specified as follows:

1.   Select *Model > Settings* from the menu,

2.   Select the 3D Slip Surface tab,

3.   Select the *Search Method* as Fully Specified - General Surface

4.   Select the *Calculation Methods* tab from the dialog and select the method types as shown below:

   Bishop Simplified

   Janbu Simplified

   Spencer

   Morgenstern-Price

5.   Select the *Convergence* tab from the menu,

   Note the number of slices: 50

6.   Press OK to close the dialog.

b. Enter Geometry (Model > Geometry)

Model geometry is defined as a set of regions and a series of layers. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from either .DXF files or from existing models.

This model will consist of a single region named Slope. To add the region follow these steps:

1.   Open the *Regions* dialog by selecting *Model > Geometry > Regions...* from the menu,

2.   Change the first region name from R1 to Slope. To do this, highlight the name and type the new text,

3.   Click *OK* to close the dialog.

The shapes that define the region will now be created. The geometry points for the Slope region are given in the table below.

•   **Define the Slope region**

1.   Select "Slope" in the Region Selector, found in the toolbar at the top of the workspace,

2.   Ensure the model view is set to 2D bye selecting View > *Mode > 2D* from the

menu,

3. Select *Draw > Geometry > Region Polygon* from the menu,

4. Move the cursor near (0,0) in the drawing space. You can view the coordinates of the mouse's current position in the status bar just below the drawing space. The SNAP and GRID options in the status bar must both be On; OSNAP should be Off,

5. To select the point as part of the shape left click on the point,

6. Now move the cursor near (680,0) and left click on the point. A line is now drawn from (0,0) to (680,0),

7. Repeat the process for the coordinate: (680,500),

8. For the final point, (0,500), double-click on the point to finish the shape. The shape is automatically finished by SVFLUX by drawing a line from (0,500) back to the start point, (0,0),

X (m)	Y (m)
0	0
680	0
680	500
0	500

N O T E :
If a mistake was made entering the coordinate points for a shape, edit the shape using the *Region Properties* dialog (menu item *Model > Geometry > Region Properties*). At times it may be tricky to snap to a grid point that is near a line defined for a region. Turn the object snap off by clicking on "OSNAP" in the status bar to alleviate this problem.

This model consists of three surfaces. By default every model initially has two surfaces.

- **Define Surface 1**
  This surface will be defined by providing a constant elevation.

    1. Select "Surface 1" in the Surface Selector found at the top of the workspace,

    2. Select *Model > Geometry > Surface Properties...* from the menu to open the *Surface Properties* dialog,

    3. For the Surface Definition Option, select Constant from the drop-down,

    4. Click on the Constant tab,

    5. Enter a Surface Constant of 50,

    6. Click *OK* to close the dialog,

- **Define Surface 2**
This surface will be defined by providing a grid of (*X,Y*) points and corresponding elevations.

7. Select "Surface 2" in the Surface Selector,

8. Go to *Model > Geometry > Surface Properties...* in the menu to open the *Surface Properties* dialog,

9. Select "Elevation Data" from the Definition Options drop-down and click the *Paste Data Grid...* button to define the grid and elevations for Surface 2,

10. Copy the (X,Y,Z) data grid for Surface 2 found in the csv file,

11. Click the *Paste Points* button,

12. Click *OK* to close the *Paste Data Grid* dialog,

13. Click *No* when asked to keep the existing grid points,

14. Click *OK* to close the *Surface Properties* dialog,

15. Click OK to close the S*urfaces* dialog.

N O T E :

You can define a custom grid by using the *Define Gridlines...* button on the *Surface Properties* dialog and entering either a set of regular or irregular gridlines. The elevation for each grid point defined by the gridlines must then be entered manually.

- **Define Surface 3**

This surface will be defined by providing a grid of (*X,Y*) points and corresponding elevations. To create the surface

16. Go to *Model > Geometry > Surfaces...* in the menu to open the *Surfaces* dialog,

17. Click the New... button to create a new surface,

18. Click *OK* on the *Insert Surfaces* dialog to use the default surface settings,

19. Click OK to close the Surfaces dialog,

20. Repeat the same steps used to define Surface 2 for Surface 3 using the (X,Y,Z) data grid found in the csv file.

d. Specify Pore Water Pressure (Model > Pore Water Pressure)

Pore water pressure is often defined in terms of a water surface and that is the case in this example. In order to specify that a water surface will be entered the user must perform the following steps:

1. Select Model > *Pore Water Pressure* > Settings...,

2. Select "Water Surfaces" as the Pore-Water Pressure Method,

3. Press *OK* to close the dialog.

The user must then proceed to enter the water surface coordinates. The water surface is defined in the same way as Surface 2 and Surface 3 were defined in the geometry section above:

4. Select *Model > Pore Water Pressure > Water Surface...*,

5. Select "Elevation Data" from the Definition Options drop-down and on the *Elevations* tab click the *Paste Data Grid...* button to define the grid and

elevations for the water surface,

6. Copy the (X,Y,Z) data grid for the water surface found in the csv file,

7. Click the *Paste Points* button,

8. Click *OK* to close the *Paste Data Grid* dialog,

9. Click *No* when asked to keep the existing grid points,

10. On the *Apply* tab, ensure that the water surface has been applied to all surfaces by clicking the *Select All* button,

11. Click *OK* to close the *Water Table Properties* dialog.

e. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the two materials that will be used in the model. The *Slope* region cuts through all the surfaces in a model, creating a separate "block" on each layer. Each block can be assigned a material or be left as void. A void area is essentially air space. In this model all blocks will be assigned a material. There are 3 surfaces resulting in two layers. This section will provide instructions on creating the *WasteRock* material. Repeat the process to add the *GlacialTill* material.

1. Open the *Materials* dialog by selecting *Model > Materials > Manager...* from the menu,

2. Click the *New...* button to create a material,

3. Enter "*WasteRock*" for the material name in the dialog that appears and choose *Mohr Coulomb* for the Shear Strength type of this material,

4. Press *OK* to close the dialog. The Material Properties dialog will open automatically,

5. Move to the Shear Strength tab and enter the parameters provided in the table below,

6. Move to the Water Parameters tab,

7. Select "On" for the Water Surfaces,

8. Click the *OK* button to close the Mohr Coulomb dialog,

9. Repeat these steps to create the *GlacialTill* material,

10. Press the *OK* button on the *Materials Manager* dialog to accept the changes and close the dialog.

Material	Shear Strength Type	Cohesion (kPa)	Friction Angle (deg)	Unit Weight (kN/m^3)	Water Surfaces
WasteRock	Mohr Coulomb	100	45	26	On
GlacialTill	Mohr Coulomb	0	35	22	On

Each region will cut through all the layers in a model, creating a separate "block" on each layer. Each block can be assigned a material or be left as void. A void area is essentially air space. In this model all blocks will be assigned a material.

1. Select *Model > Materials > Material Layers...* from the menu to open the

Material Layers dialog,

2. Select "GlacialTill" from the drop-down for Layer 2 and "WasteRock" from the drop-down for Layer 1,

3. Close the dialog using the *OK* button.

f.   Specify Search Method Geometry (Model > Slip Surface)

The General Sliding Surface method of searching for the critical slip surface has already been selected in Step b. Now the user must enter the data grid for the surface. This is accomplished through the following steps:

1. Select *Model > Slip Surface > Fully Specified > General Surface...*,

2. Click the *Paste Data Grid...* button to define the grid and elevations for the general sliding surface,

3. Copy the (X,Y,Z) data grid for the surface found in the csv file,

4. Click the *Paste Points* button,

5. Click *OK* to close the *Paste Data Grid* dialog,

6. Click *OK* to close the *General Sliding Surface* dialog.

The General Sliding Surface graphics should now be displayed on the CAD window.

h.   Run Model (Solve > Analyze)

The next step is to analyze the model.

1. Select *Solve > Analyze* from the menu. A pop-up dialog will appear and the solver will start.

### i.   Visualize Results (Window > AcuMesh)

After the model has completed solving, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on the *Acumesh...* button. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen.

14.2   Results and Discussion

After the model has completed solving the user may view the results in the ACUMESH software by pressing the ACUMESH icon on the process toolbar. The sliding mass is displayed in the CAD for the selected calculation method. To switch between the results of the different calculation methods, click on the drop down menu at the top of the screen and select the method you would like to view. In order to view the sliding mass area more clearly the user may edit the *Critical Sliding Mass* dialog:

1. Select *Slips > Critical Sliding Mass...* from the menu,

2. Under the *Mass Explosion* tab check the *Show Sliding Mass with Explosion under 3D View* checkbox,

3. Move the Explosion distance slider to the right.

The analysis results in a factor of safety of 2.186 for the Morgenstern-Price method.
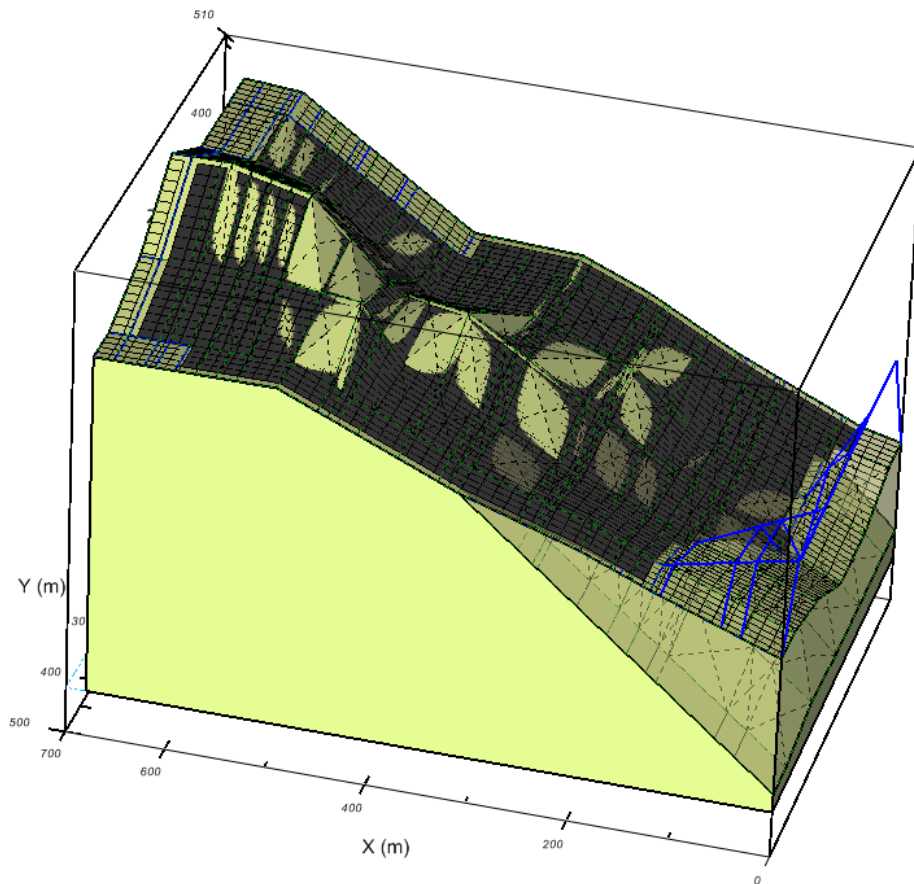
15   2D Rapid Drawdown Example

Water tables placed against earth levees over time will adjust to steady state conditions. If the water level against the earth levee is suddenly lowered then pore-water pressures in the earth levee may not dissipate fast enough and can lead to a slope failure situation. The following example is used to illustrate the use of the total stress method (Duncan three-stage) for rapid drawdown analysis of a two-dimensional storage dam model. The purpose of this model is to document the correct solution of the rapid drawdown methodology as presented by Duncan et al. (1990).
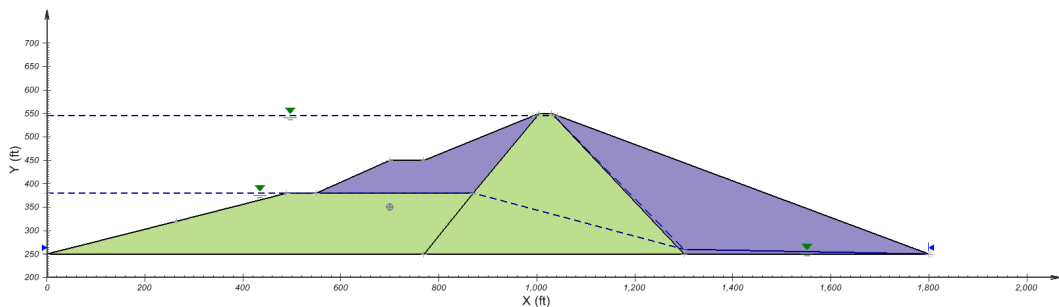
This original model can be found under:

Project: Slopes_Group_3
Model: RDD_Pumped_Storage_Project_Dam

Minimum authorization required: PROFESSIONAL

Model Description and Geometry

The pumped storage project dam has a densely compacted, silty clay core. The lower portion of the upstream slope is a random zone with the same strength properties as the core. The upper portion of the upstream slope and all of the downstream slope is a free draining rockfill. The rapid drawdown analysis water level is from 545 feet to 380 feet.

15.1   Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the following general categories:

      a.   Create model

      b.   Specify analysis settings

      c.   Enter geometry

      d.   Specify pore water pressure

      e.   Apply material properties

      f.   Specify search method geometry

      g.   Run model

      h.   Visualize results

The details of these outlined steps are given in the following sections.

NOTE:

Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

Since PROFESSIONAL authorization is required for this tutorial, perform the following steps to ensure PROFESSIONAL authorization is activated:

1. Plug in the USB security key,

2. Go to the *File > Authorization* dialog on the SVOFFICE Manager, and

3. Software should display PROFESSIONAL authorization. If not, it means that the security codes provided by SoilVision Systems at the time of purchase have not yet been entered. Please see the Authorization section of the SVOFFICE User's Manual for instructions on entering these codes.

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,

2. Press the *Clear Search* button (if enabled),

3. Create a new project called "UserTutorial" by pressing the *New...* button next to the list of projects,

4. Create a new SVSlope model called "Rapid Drawdown Example 2D" by pressing the SVSlope icon located above the list of models. Use the settings below when creating this new model:

> Application: SVSlope
> Model Name: Rapid Drawdown Example 2D
> System: 2D
> Units: Imperial
> Slip Direction: Right to Left

5. Click on the *World Coordinate System* tab and enter the World Coordinates System coordinates shown below,

> x min = 0   x max = 2000
> y min = 0   y max = 1000

6. Click on *OK*.

The new model will be automatically added under the recently created UserTutorial project.

SVSLOPE now opens to show a grid and the Options dialog (*View > Options)* pops up. Change the default horizontal and vertical grid spacing to 5 ft and click *OK*.

b. Specify Analysis Settings (Model > Settings)

In SVSlope the analysis settings provide the information for what type of analysis will be performed. These settings will be specified as follows:

1.   Select *Model > Settings* from the menu,
2.   Select the Slip Surface tab,

> Slip Direction: Right to Left
>
> Slip Shape: Circular
>
> Search Method: Grid and Tangent

3.   Select the *Calculation Methods* tab from the dialog and select the method types as shown below:

> Bishop Simplified
>
> Spencer
>
> Corps of Engineers #2
>
> Morgenstern-Price
>
> GLE (Fredlund)
>
> Sarma

4.   Select the *Applications* tab from the dialog,
5.   Check the "Apply Rapid Drawdown Analysis" check box,
6.   Press OK to close the dialog.

c. **Enter Geometry** (Model > Geometry)

Model geometry is defined as a set of regions. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from either .DXF files or from existing models.

The shapes that define each region will now be created. Refer to the data at the end of this tutorial for the geometry points for the four regions.

- **Define Region R1**

1.   Select *Draw > Geometry > Polygon Region* from the menu,
2.   The cursor will now be changed to a cross hair,
3.   Move the cursor near (1005,550) in the drawing space,
4.   To select the point as part of the shape left click on the point,
5.   Now move the cursor near (870,380) and left-click the mouse. A line is now drawn from (1005,550) to (870,380),
6.   In the same manner then enter the following points:
     (550,380)
     (700,450)
7.   Move the cursor near the point (770,450). Double click on the point to finish the shape. A line is now drawn from (700,450) to (770,450). The shape is automatically finished by SVSLOPE by drawing a line from (770,450) back to the start point (10005,550).

Repeat this process to define the regions *R2*, *R3* and *R4*.

> **N O T E :**

If an error is made when entering the region geometry the user may recover from the error and start again by one of the following methods:
   a. Press the escape (esc) key.
   b. Select a region shape and press the delete key.
   c. Use the Undo function on the Edit menu.

d. Specify Pore Water Pressure (Model > Pore Water Pressure)

Initial and final conditions are used to model the rapid drawdown of the water table. To enter the two water tables into SVSLOPE perform the following steps:

1. Select *Model > Pore Water Pressure > Settings...*,

2. Select "Water Surfaces" as the Pore-Water Pressure Method,

3. Press *OK* to close the dialog.

The user must then proceed to enter the initial water table coordinates:

1. Select *Model > Pore Water Pressure > Water Table...*,

2. Under the *Points* tab enter the *X* and *Y* coordinates as provided for the initial water table at end of this tutorial,

3. Press *OK* to close the dialog.

The user must then proceed to enter the final water table coordinates:

4. Select *Model > Final Conditions > Water Table...*,

5. Under the *Points* tab enter the *X* and *Y* coordinates as provided for the final water table at end of this tutorial,

6. Press *OK* to close the dialog.

e. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the two materials that will be used in the model.

1. Open the *Materials Manager* dialog by selecting *Model > Materials > Manager...* from the menu,

2. Click the *New* button to create a material,

3. Enter "Silty Clay Core" for the material name in the dialog that appears and choose Mohr Coulomb for the Shear Strength type of this material,

4. Press *OK* to close the dialog. The *Material Properties* dialog will open automatically,

N O T E :
When a new material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned to it will display that material's fill color.

5. Move to the *Shear Strength* tab,

6. Enter the Unit Weight value of 140 lb/ft³,

7. Enter the Cohesion, c value of 0 psf,

8. Enter the Friction Angle, phi value of 36 degrees,

9. Check the "Apply Rapid Drawdown" checkbox,

10. Enter the Cohesion, cT value of 2000 psf,

11. Enter the Friction Angle, phiT of 18 deg,

12. Press *OK* button to close the *Material Properties* dialog,

13. Repeat these steps to create the "compacted Rockfill" material using the information provided in the table below.

Material	Shear Strength Type	Cohesion (psf)	Friction Angle phi (deg)	Unit Weight (lb/ft³)	Cohesion cT (psf)	Friction Angle phiT (deg)
Silty Clay Core	Mohr Coulomb	0	36	140	2000	18
compacted Rockfill	Mohr Coulomb	0	37	142	N/A	N/A

Once all material properties have been entered, we must apply the materials to the corresponding regions.

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions...* from the menu,

2. Assign the compacted Rockfill material to region R1 using the drop down,

3. Assign the Silty Clay Core material to region R2 using the drop down,

4. Assign the Silty Clay Core material to region R3 using the drop down,

5. Assign the compacted Rockfill material to region R4 using the drop down,

6. Press the OK button to accept the changes and close the dialog.

f. Specify Search Method Geometry (Model > Slip Surface)

The Grid and Tangent method of searching for the critical slip surface has already been selected in Step b. Now the user must specify the exact geometry of these objects. This is accomplished through the following steps:

GRID

1. Select *Model > Slip Surface > Grid and Tangent...*,

2. Select the *Grid* tab,

3. Enter the values for the grid as specified at the end of this tutorial,

4. Move to entering the tangent values.

TANGENT

1. Select the *Tangent* tab,

2. Enter the values for the tangent as specified at the end of this tutorial,

3. Click *OK* to close the dialog,

The grid and tangent graphics should now be displayed on the CAD window.

h. Run Model (Solve > Analyze)

The next step is to analyze the model.

> 1.   Select *Solve > Analyze* from the menu. A pop-up dialog will appear and the solver will start.

i. Visualize Results (Window > AcuMesh)

After the model has completed solving, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on the *Acumesh...* button. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen.

15.2   Results and Discussion

After the model has completed solving the user may view the results in the ACUMESH software by pressing the ACUMESH icon on the process toolbar. The sliding mass is displayed in the CAD for the selected calculation method. To switch between the results of the different calculation methods, click on the drop down menu at the top of the screen and select the method you would like to view. The analysis results in a factor of safety of 1.504 for Spencer's method. The critical slip surface for Spencer's method is shown in the following screenshot.
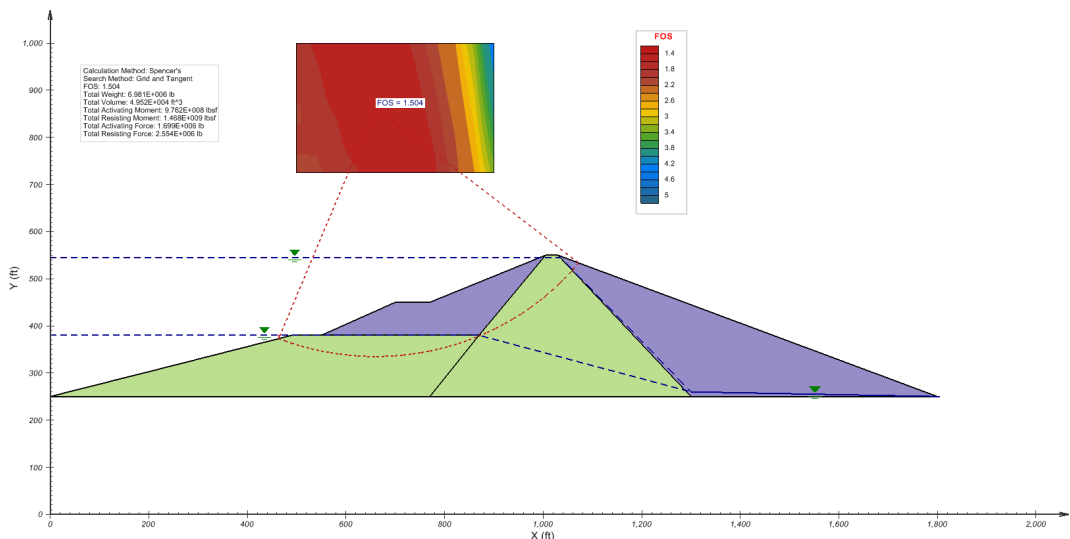
15.3   Model Data

Region Geometries

Region: R1

X (ft)	Y (ft)
1005	550
870	380
550	380
700	450
770	450

Region: R2

X (ft)	Y (ft)
870	380
770	250
0	250
265	320
490	380
550	380

Region: R3

X (ft)	Y (ft)
1005	550
1030	550
1300	250
770	250
870	380

Region: R4

X (ft)	Y (ft)
1030	550
1800	250
1300	250

Pore-Water Pressure

Initial Water Table:

X (ft)	Y (ft)

0	545
1035	545
1300	260
1800	250

Final Water Table:

X (ft)	Y (ft)
0	380
870	380
1300	260
1805	250

Grid and Tangent

Grid - Points

	X	Y
Upper Left	500	1000
Lower Left	500	725
Lower Right	900	725

X increments		10
Y increments		10

Tangent - Points

	X	Y
Upper Left	0	380
Lower Left	0	320
Lower Right	1000	320
Upper Right	1000	380

Increments	5

16   3D Rapid Drawdown Example

The following example is used to illustrate the analysis of the 2D Rapid Drawdown Example tutorial model extended to a three-dimensional model. The purpose of this model is to compare the factor of safety to that found in the two-dimensional version of this model.

This original model can be found under:

Project: Slopes_3D
Model: RDD_Pumped_Storage_Project_Dam_3D

Minimum authorization required: PROFESSIONAL

Model Description and Geometry

This model extends the 2D Rapid Drawdown Example tutorial model into three-dimensions by using a width of 1000 ft. All other aspects of this model are the same as those found in the two-dimensional version.
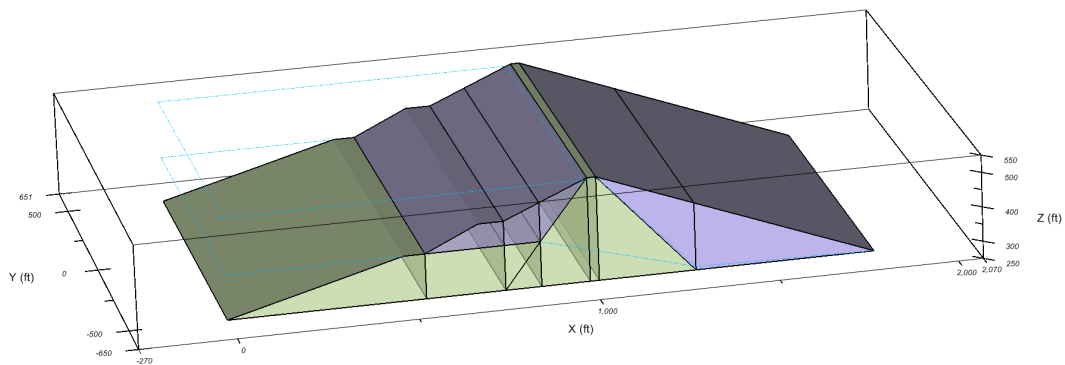
16.1   Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the following general categories:

   a.   Open 2D Rapid Drawdown Example tutorial model

   b.   Extrude 2D Model to 3D

   c.   Run model

   d.   Visualize results

The details of these outlined steps are given in the following sections.

> **N O T E :**
> Any values on the dialogs that are not specifically mentioned in the steps below are
> assumed to be the default values currently present.

a.   Open 2D Rapid Drawdown Example Tutorial Model

This model begins with a two-dimensional version. To open the model follow these steps:

1. Open the SVOFFICE Manager dialog,
2. Select the project where the 2D Rapid Drawdown Example tutorial model was created,
3. Locate the the 2D Rapid Drawdown Example tutorial model in the list of models and open the model,

> **N O T E :**
> The RDD_Pumped_Storage_Project_Dam model located in the Slopes_Group_3 project may be utilized as the two-dimensional model referenced above if the model in the 2D Rapid Drawdown Example tutorial has not been created. However, the Grid and Tangent slip search method will need to be specified. The steps to specify this search method are described in the Model Setup section of the 2D Rapid Drawdown Example.

b. Extrude 2D Model to 3D (File > Save As)

A new model is created with 3D geometry by extruding the 2D cross-section from the current model. This is accomplished through the following steps:

1. First, save the current model by clicking *File > Save* from the menu,
2. Next, to begin the extrusion process select *File > Save As...* from the menu,
2. Select the *General* tab,

> System: 3D
>
> New File Name: Rapid Drawdown Example 3D

3. Select the *Spatial* tab,
4. Enter the following model extrusion parameters,

> Y minimum: -500 ft
>
> Y maximum: 500 ft

5. Press *OK* to close the dialog,
6. Press *OK* to accept the reset of some items.
7. Select *View > Mode > 3D* to change the CAD to a 3D view.

> **N O T E :**
> X- and Y-coordinates in 2D become X- and Z-coordinates in 3D space with the model extrusion

The Y-coordinates for the search method geometry needs to be updated in the 3D model:

8.   Select *Model > Slip Surface > Grid and Tangent...*,
9.   Enter the following values for the Y-coordinate,

> Min Value: 0.5
>
> Max Value: 0.5
>
> No. of Points: 1

The slope limits also need to be reset in the new 3D geometry:

10. Select *Model > Slope Limits*,

> Min x: 0 ft   Max x: 1800 ft
>
> Min y: -500 ft   Max y: 500 ft

11. Press *OK* to close the dialog.

The 3D model is now complete and ready to be analyzed.

c. Run Model (Solve > Analyze)

The next step is to analyze the model.

1.   Select *Solve > Analyze* from the menu. A pop-up dialog will appear and the solver will start.

d. Visualize Results (Window > AcuMesh)

After the model has completed solving, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on the *Acumesh...* button. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen.

16.2   Results and Discussion

After the model has completed solving the user may view the results in the ACUMESH software by pressing the ACUMESH icon on the process toolbar. The sliding mass is displayed in the CAD for the selected calculation method. To switch between the results of the different calculation methods, click on the drop down menu at the top of the screen and select the method you would like to view. In order to view the sliding mass area more clearly the user may edit the *Critical Sliding Mass* dialog:

1. Select *Slips > Critical Sliding Mass...* from the menu,
2. Adjust the *Explosion Distance* slider to see the 3D shape of the critical sliding mass.

The analysis results in a factor of safety of 1.766 for Spencer's method.
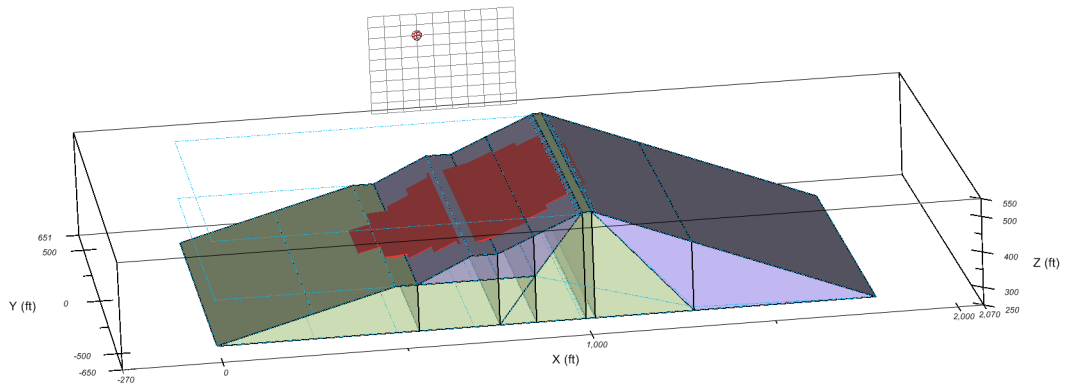
The factor of safety is slightly higher than that found in the two-dimensional version of this model (2D Rapid Drawdown Example). This leads to the conclusion that adding the width dimension to the two-dimensional model yields in a slightly more stable slope.

17   3D Arbitrary Sliding Direction

This example is used to illustrate the analysis of a three-dimensional slope stability model using the Multi-Directional Slip Analysis feature of SVSLOPE, i.e., a slip surface direction that does not follow the x-axis. A range of slip surface directions is analyzed and the effect on the factor of safety for the slope is noted.

This example consists of a simple one layer slope. The model is analyzed using the Bishop Simplified method and the GLE method. The purpose of this example is to analyze the stability of a simple slope along several different slip surface directions and present the resultant factors of safety.
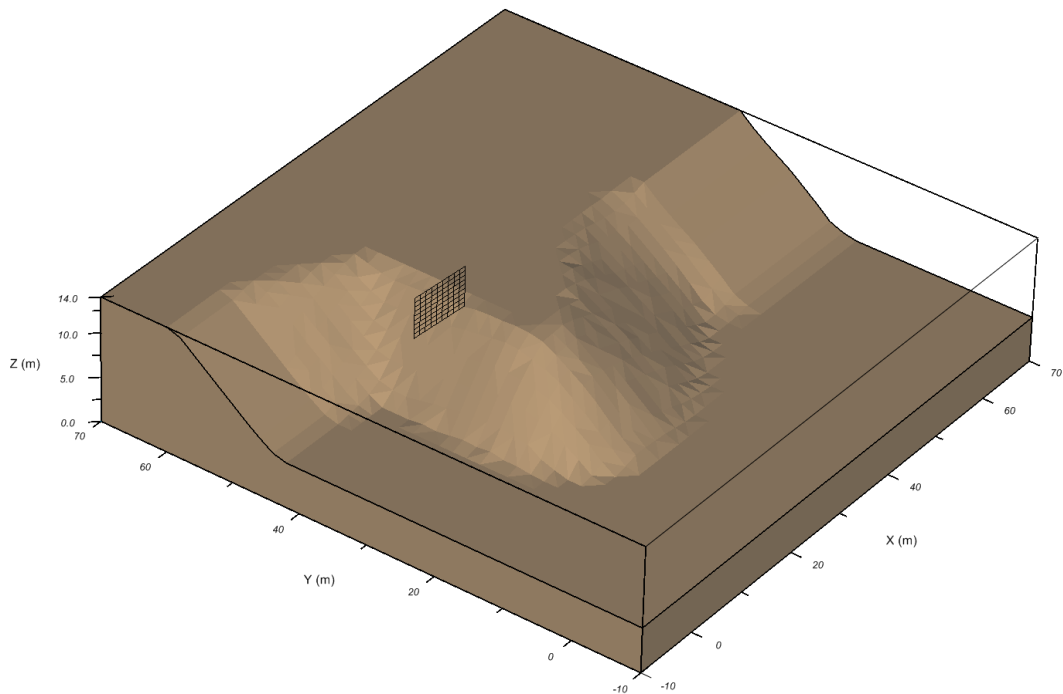
The model is developed from: Jiang, Can. Geotech. J. 40: 308-325 (2003). Jiang results were a FOS = 1.33 using the Dynamic Programming search method and the Janbu analysis method.

This original model can be found under:

Project: Slopes_3D
Model: Arbitrary_Sliding_Direction

Minimum authorization required to complete this tutorial: ELITE

Model Geometry

17.1   Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

    a.   Create model

    b.   Enter geometry

    c.   Specify analysis settings

    d.   Specify search method geometry

    e.   Apply material properties

    f.   Run model

    g.   Visualize results

The details of these outlined steps are given in the following sections.

> **N O T E :**
> Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a.   Create Model

ELITE authorization is required for this tutorial. The steps to ensure that ELITE authorization is activated are described in the Authorization section.

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,

2. Press the Clear Search button (if enabled),

3. Select an existing project or create a new project called "UserTutorial" by pressing the "Create New Project" icon found above the list of projects,

4. Create a new model called "Arbitrary Sliding Direction" by pressing the SVSlope button above the list of models. Use the settings below when creating this new model:

      Application: SVSLOPE

      Model Name: ASD

      System: 3D

      Units: Metric

      Slip Direction: Multi-Directional

5. Click on the *World Coordinate System* tab and enter the World Coordinates System coordinates shown below,

      x min = -10   x max = 70

      y min = -10   y max = 70

      z min = 0   z max = 14

6. Click on *OK*.

The new model will be automatically added under the recently created UserTutorial project or the previously selected project.

SVSLOPE now opens to show a grid and the *Options* dialog (*View > Options)* pops up. Click *OK* to accept the default horizontal and vertical grid spacing.

b. Enter Geometry (Model > Geometry)

Model geometry is defined as a set of regions and a series of layers. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from either .DXF files or from existing models.

This model contains a single region. Every model has one region defined by default. The shape that defines this region will now be created.

- **Define R1 Region**

    1. Select *Model > Geometry > Region Properties...* from the menu,
    2. Click the *New Polygon* button,
    3. Copy and paste the region coordinates from the table below into the *New Polygon Shape* dialog using the *Paste Points* button (do not copy the header row),
    4. Press *OK* to close the dialogs.

Region: R1

X (m)	Y (m)
-10	-10
70	-10
70	70
-10	70

This model consists of two surfaces. By default every model initially has two surfaces.

- **Define Surface 1**

This surface will be defined by providing a constant elevation.

    1. Select *Model > Geometry > Surfaces...* from the menu to open the *Surfaces* dialog,
    2. Select the row containing "Surface 1" in the surface list and click the *Properties...* button,
    3. For the Surface Definition Option, select Constant from the drop-down,
    4. Click on the Constant tab,
    5. Enter an Elevation of 0,
    6. Click *OK* to close the dialog.

- **Define Surface 2**

This surface will be defined by providing a regular grid of X and Y grid lines and corresponding elevations.

1. Select the row containing "Surface 2" in the surface list and click the *Properties...* button,
2. Select "Elevation Data" from the Definition Options,
3. Click the *Paste Data Grid...* button to set up the grid for the selected surface,
4. Copy the (X,Y,Z) data grid for Surface 2 found in the csv file 3D Arbitrary Sliding Direction Data for Surface 2 (do not include the header information),
5. Click the *Paste Points* button on the *Paste Data Grid dialog*,
6. Click *OK* to close the *Paste Data Grid* dialog,
7. Click OK to close the *Surface Properties* dialog,
8. Click OK to close the *Surfaces* dialog,

If all model geometry has been entered correctly the shape should look like the diagram at the start of this tutorial.

c.   Specify Analysis Settings (Model > Settings)

In SVSlope the Analysis Settings provide the information for what model output will be available in ACUMESH. These settings are specified as follows:

1. Select *Model > Settings...* from the menu,
2. Move to the *Calculation Methods* tab and select the method type as shown below:

    Bishop Simplified
    GLE

3. Move to the Multi-Directional Slip Analysis tab and click the *Draw...* button to draw the primary slip direction on the CAD,
4. Move the cursor to the point (26, 32) and left-click,
5. Move the cursor to the point (-4, 32) and click to complete the slip direction,
6. Enter the following Rotation Angles by clicking the *Add Regular...* button,

Start:	-10
Increment value:	10
Number of Increments:	3
End:	10

7. Move to the Convergence tab and enter the values as follows. Note that a coarser column grid and reduced number of slices are defined in order to decrease the model solving time. These modifications were found to have little effect on the factor of safety compared to the default values.

Number of rows (Y direction):	40
Number of slices:	40

Tolerance: 0.001

Maximum number of iterations: 50

8. Press *OK* to close the dialogs.

d. Specify Search Method Geometry

The Grid and Tangent method of searching for the critical slip surface has already been selected in a previous step. Now the user must specify the geometry for each of these objects. This is accomplished through the following steps:

GRID

1.   Select *Model > Slip Surface > Grid and Tangent...*,

2.   Select the *Grid and Tangent* tab,

3.   Enter the values provided in the table below,

4.   Press *OK* to close the dialog.

	Min Value	Max Value	Points
X* Coordinate	2	10	5
Y* Coordinate	32	32	1
Z Coordinate	15	20	5
Tangent Planes	4.691	5.146	4
Aspect Ratio	0.5	1	6

N O T E :
The X* and Y* coordinates are rotated coordinates, i.e., they are relative to the slip direction.

The grid and tangent graphics should now be displayed on the CAD window.

e. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material property for the material that will be used in the model. This section will provide instructions on creating the *soil* material.

1.   Open the *Materials* dialog by selecting *Model > Materials > Manager...* from the menu,

2.   Click the *New...* button to create a material,

3.   Enter "soil" for the material name in the dialog that appears and choose Mohr Coulomb for the Shear Strength type of this material,

4.   Press *OK* to close the dialog. The *Material Properties* dialog will open automatically,

N O T E :

When a new material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned to it will display that material's fill color.

5. Move to the *Shear Strength* tab and enter the parameter values given in the table below,

6. Click the *OK* button to close the *Shear Material Properties* dialog,

Material	Shear Strength Type	Cohesion (kPa)	Friction Angle phi (deg)	Unit Weight (kN/m³)
soil	Mohr Coulomb	11.7	24.7	17.66

Once the material property has been entered, we must apply the material to the appropriate region. Each region will cut through all the layers in a model, creating a separate "block" on each layer. Each block can be assigned a material or be left as void. In this model there is only one region and one layer. The material is assigned to this block as follows.

1. Open the *Material Layers* dialog by selecting *Model > Materials > Material Layers...* from the menu,

2. Select the *soil* Material for Layer 1 from the drop down,

3. Press the *OK button* to close the dialog.

f.   Run Model (Solve > Analyze)

The next step is to analyze the model. SVSlope will automatically iterate through each of the slip direction angles defined above. The current slip direction angle is shown in the *Search Method* geometry description.

1. Select *Solve > Analyze* from the menu. A pop-up dialog will appear and the solver will start,

g. Visualize Results (Window > AcuMesh)

After the model has been run, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on *Yes*. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH as appears in the diagrams following. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen. To switch between the results of the different methods selected, click on the drop down menu at the top of the workspace and select the method you would like to view.

17.2   Results and Discussion

The sliding mass is displayed in the CAD for the selected calculation method. In order to view the sliding mass area more clearly the user may edit the *Critical Sliding Mass* dialog:

1. Select *Slips > Critical Sliding Mass...* from the menu,

2. Move to the Mass Explosion tab and adjust the *Explosion Distance* slider.
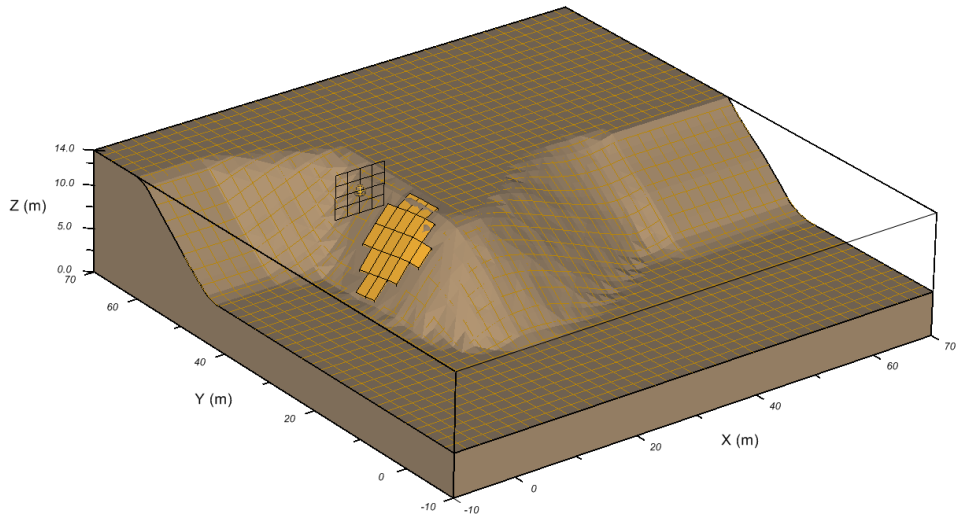
The analysis results in a factor of safety of 1.404 along the 0 degree sliding direction angle for the Bishop's method. A screenshot of the two-dimensional view along the sliding direction is shown below. This view is accessed by clicking on the SD icon which appears below the toolbars on the top left hand side of the screen
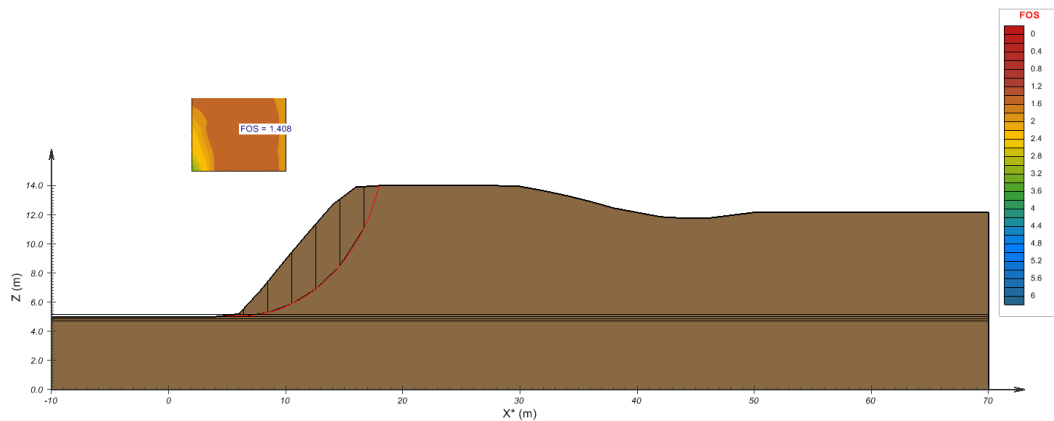
The factor of safety versus sliding direction angle is shown below. The dialog is available by clicking *Slips > FOS vs Sliding Direction Angle...* in the menu. As displayed in the screenshot below, the angle with the lowest factor of safety is the 0 degree slip direction angle.
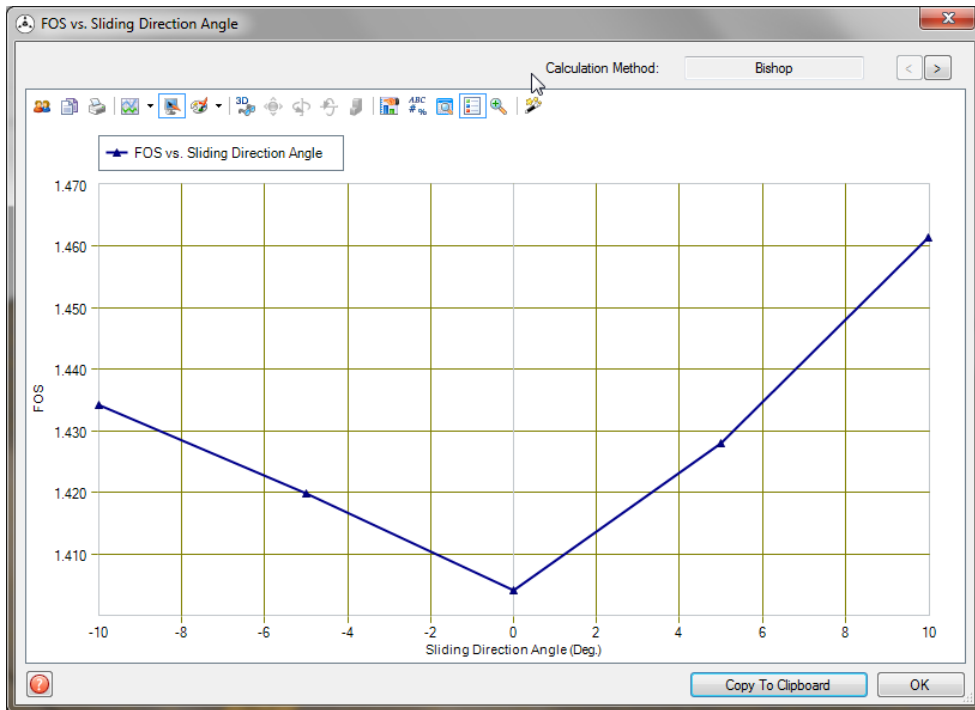
18   3D Open Pit Analysis

This example is used to illustrate the analysis of a three-dimensional slope stability model for the slope of an open pit The slip direction is taken as parallel to the x-axis. A assumed fault is input into the software. The searching for the slip surface uses a combination of elliptical entry and exit slip surfaces as well as intersection with a fault.

This original model can be found under:

Project: Slopes_3D
Model: Open_Pit_LEM_32_Fast

Minimum authorization required to complete this tutorial: PROFESSIONAL
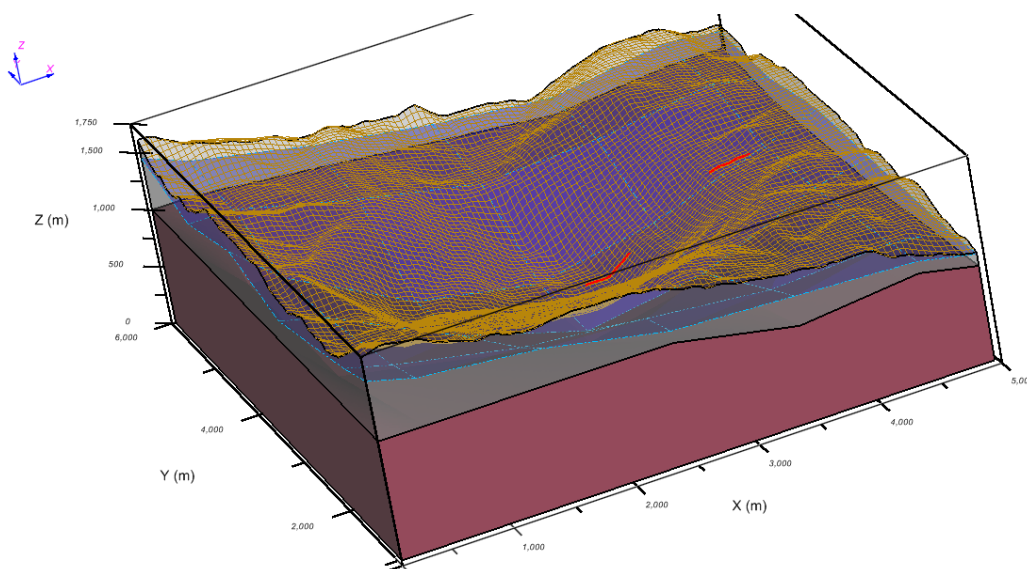
Model Geometry

18.1   Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

       a.   Create model

       b.   Enter geometry

       c.   Specify analysis settings

       d.   Specify search method geometry

       e.   Apply material properties

       f.   Run model

       g.   Visualize results

The details of these outlined steps are given in the following sections.

NOTE:
Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

FULL authorization is required for this tutorial. The steps to ensure that full authorization is activated are described in the Authorization section.

The following steps are required to create the model:

1.   Open the *SVOFFICE Manager* dialog,

2.   Press the Clear Filter button (if enabled),

3.   Select an existing project or create a new project called "UserTutorial" by pressing the "Create New Project" icon found above the list of projects,

4.   Create a new model called "Arbitrary Sliding Direction" by pressing the SVSlope button above the list of models. Use the settings below when creating this new model:

>   Application: SVSLOPE
>   Model Name: Open_Pit
>   System: 3D
>   Units: Metric
>   Slip Direction: Towards negative X

5.   Click on the *World Coordinate System* tab and enter the World Coordinates System coordinates shown below,

>   x min = 100   x max = 5000
>   y min = 900   y max = 6000
>   z min = 0   z max = 1750

6.   Click on *OK*.

The new model will be automatically added under the recently created UserTutorial project or the previously selected project.

SVSLOPE now opens to show a grid and the *Options* dialog (*View > Options)* pops up. Click *OK* to accept the default horizontal and vertical grid spacing.

b. Enter Geometry (Model > Geometry)

Model geometry is defined as a series of layers and can be either drawn by the user or defined as a set of coordinates. Model Geometry can be imported from either .DXF files or from existing models.

This model contains a single region. Every model has one region defined by default. The shape that defines this region will now be created.

- **Define R1 Region**

1. Select *Model > Geometry > Region Properties...* from the menu,
2. Click the *New Polygon* button,
3. Copy and paste the region coordinates from the table below into the *New Polygon Shape* dialog using the *Paste Points* button (do not copy the header row),
4. Press *OK* to close the dialogs.

Region: R1

X (m)	Y (m)
127	983
127	5982
4976.3	5982
4976.3	983

This model consists of two surfaces. By default every model initially has two surfaces.

- **Define Surface 1**

This surface will be defined by providing a constant elevation.

1. Select *Model > Geometry > Surfaces...* from the menu to open the *Surfaces* dialog,
2. Select the row containing "Surface 1" in the surface list and click the *Properties...* button,
3. For the Surface Definition Option, select Constant from the drop-down,
4. Click on the Constant tab,
5. Enter an Elevation of 0,
6. Click *OK* to close the dialog.

- **Define Surface 2**

This surface will be defined by providing a regular grid of X and Y grid lines and corresponding elevations.

1. Select the row containing "Surface 2" in the surface list and click the *Properties...* button,
2. Select "Elevation Data" from the Definition Options,
3. Click the *Paste Data Grid...* button to set up the grid for the selected surface,
4. Copy the (X,Y,Z) data grid for Surface 2 found in the XLS file "Open_Pit" (do not include the header information),
5. Click the *Paste Points* button on the *Paste Data Grid dialog*,
6. Click *OK* to close the *Paste Data Grid* dialog,
7. Click OK to close the *Surface Properties* dialog,
8. Click OK to close the *Surfaces* dialog,

Define Surface 3
This surface will be defined by providing a regular grid of X and Y grid lines and corresponding elevations.

9. Select the row containing "Surface 3" in the surface list and click the *Properties...* button,

10. Select "Elevation Data" from the Definition Options,

11. Click the *Paste Data Grid...* button to set up the grid for the selected surface,

12. Copy the (X,Y,Z) data grid for Surface 3 found in the XLS file "Open_Pit" (do not include the header information),

13. Click the *Paste Points* button on the *Paste Data Grid dialog*,

14. Click *OK* to close the *Paste Data Grid* dialog,

15. Click OK to close the *Surface Properties* dialog,

16. Click OK to close the *Surfaces* dialog,

If all model geometry has been entered correctly the shape should look like the diagram at the start of this tutorial.

c.   Specify Analysis Settings (Model > Settings)
In SVSlope the Analysis Settings provide the information for what model output will be available in ACUMESH. These settings are specified as follows:

1. Select *Model > Settings...* from the menu,

2. On the 3D Slip Surface tab select the "Entry and Exit with Fully Specified Wedges" and "Towards negative X" slip direction

3. Move to the *Calculation Methods* tab and select the method type as shown below:

    Bishop Simplified

4. Move to the Convergence tab and enter the values as follows. Note that a coarser column grid and reduced number of slices are defined in order to decrease the model solving time. These modifications were found to have little effect on the factor of safety compared to the default values.

Number of rows (Y direction):	100
Number of slices:	100
Tolerance:	0.001
Maximum number of iterations:	50
Minimum slide surface depth:	50
Minimum number of active columns:	40

5. Press *OK* to close the dialogs.

d. Specify Search Method Geometry

The Entry and Exit method of searching for the critical slip surface has already been selected in a previous step. Now the user must specify the geometry for each of these objects. This is accomplished through the following steps:

3D Entry and Exit

Definition | Format

Entry Range(Right Side)

◉ Range   ○ Point

Left Point		Right Point	
X:	3847.313	X:	4197.964
Z:	1016.978	Z:	1070.138

Draw   Increments: 10

Exit Range(Left Side)

◉ Range   ○ Point

Left Point		Right Point	
X:	2702	X:	3099
Z:	434.665	Z:	582.192

Draw   Increments: 10

Radius increments: 10

	Min. Value:	Max. Value:	Total Divisons
Y-Coordinate:	3482.5	3482.5	1
Aspect Ratio:	0.5	0.7	3

Apply   2D View   OK   Cancel

Fully Specified Wedge Sliding Surface

	X (m)	Y (m)	Z (m)	Dip (deg.)	Dip Dir. (Deg.)	Disc. Material
▶ Wedge #1	3863	3482.5	900	35	0	☐ weak ▾
✱						▾

Apply   2D View   Paste Points   Delete   Delete All   OK   Cancel

e. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material property for the material that will be used in the model. This section will provide instructions on creating the *soil* material.

1. Open the *Materials* dialog by selecting *Model > Materials > Manager...* from the menu,

2. Click the *New...* button to create a material,

3. Enter "soil" for the material name in the dialog that appears and choose Mohr

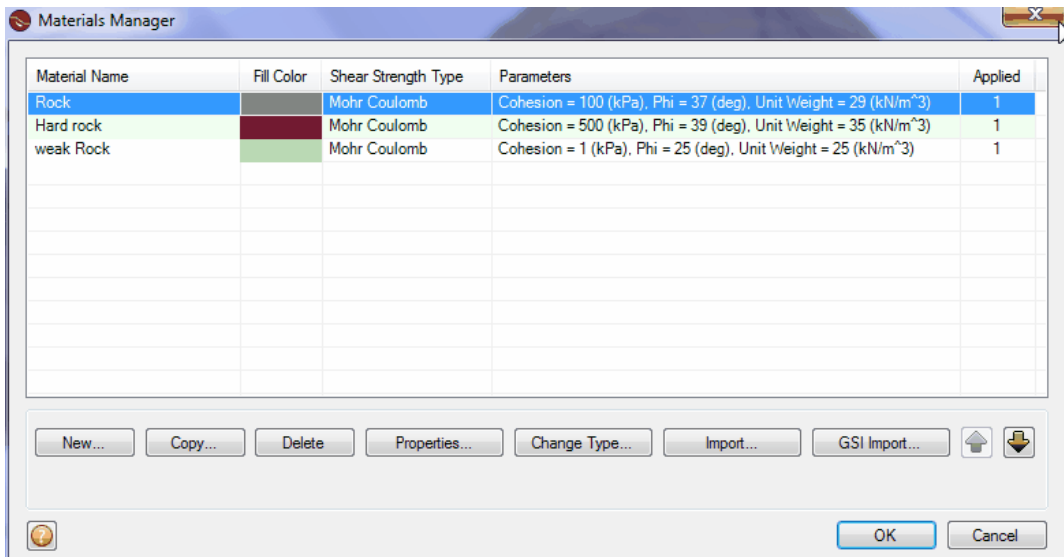Coulomb for the Shear Strength type of this material,

4. Press *OK* to close the dialog. The *Material Properties* dialog will open automatically,

---

N O T E :

When a new material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned to it will display that material's fill color.

---

5. Move to the *Shear Strength* tab and enter the parameter values given in the table below,

6. Click the *OK* button to close the *Shear Material Properties* dialog,

Once the material property has been entered, we must apply the material to the appropriate region. Each region will cut through all the layers in a model, creating a separate "block" on each layer. Each block can be assigned a material or be left as void. In this model there is only one region and one layer. The material is assigned to this block as follows.

1. Open the *Material Layers* dialog by selecting *Model > Materials > Material Layers...* from the menu,

2. Select the *soil* Material for Layer 1 from the drop down,

3. Press the *OK button* to close the dialog.

f. Run Model (Solve > Analyze)

The next step is to analyze the model. SVSlope will automatically iterate through each of the slip direction angles defined above. The current slip direction angle is shown in the *Search Method* geometry description.

1. Select *Solve > Analyze* from the menu. A pop-up dialog will appear and the solver will start,

g. Visualize Results (Window > AcuMesh)

After the model has been run, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on *Yes*. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH as appears in the diagrams following. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen. To switch between the results of the different methods selected, click on the drop down menu at the top of the workspace and select the method you would like to view.

18.2   Results and Discussion

The sliding mass is displayed in the CAD for the selected calculation method. In order to view the sliding mass area more clearly the user may edit the *Critical Sliding Mass* dialog:

1. Select *Slips > Critical Sliding Mass...* from the menu,
2. Move to the Mass Explosion tab and adjust the *Explosion Distance* slider.
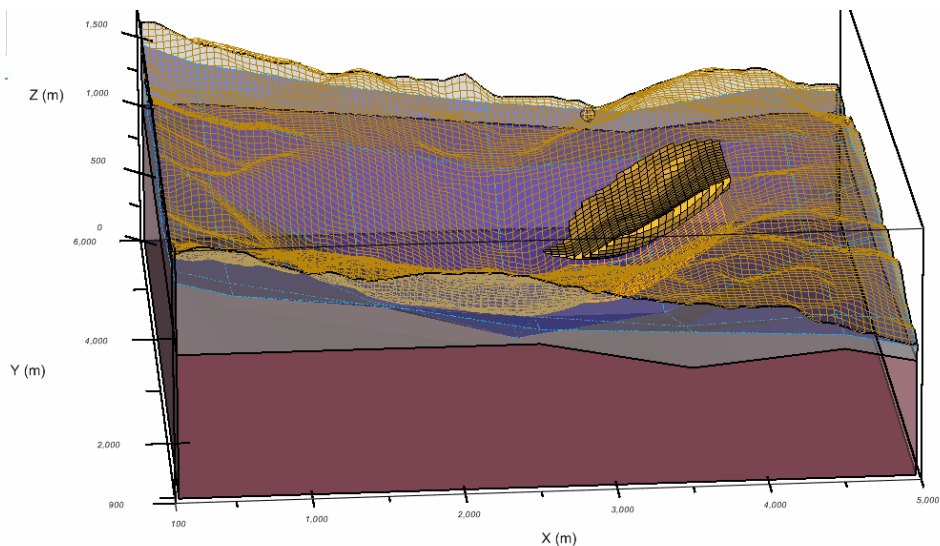
The analysis results in a factor of safety of 1.377 along the x-axis sliding direction angle for the Bishop's method. A screenshot of the two-dimensional view along the sliding direction is shown below. This view is accessed by clicking on the SD icon which appears below the toolbars on the top left hand side of the screen